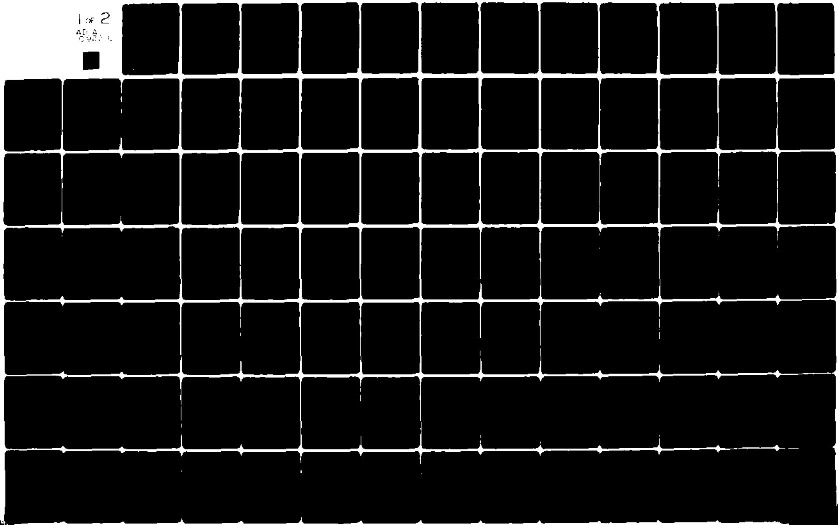
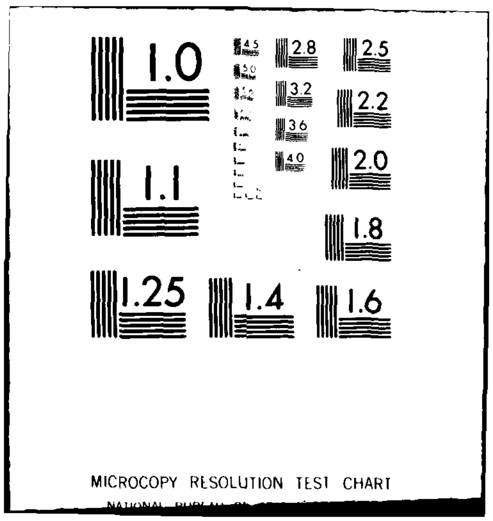
AD-A092 210 SWANSON SERVICE CORP HUNTINGTON BEACH CA F/G 13/13
EVALUATION OF THE NASTRAN GENERAL PURPOSE COMPUTER PROGRAM.(U)
AUG 80 J W JONES, H H FONG, D A BLEHM N00014-79-C-0576
UNCLASSIFIED SSC-81980 NL

1 of 2
AD A
092210




1.0
1.1
1.25
1.4
1.6
1.8
2.0
2.2
2.5
2.8
3.2
3.6
4.0
MICROCOPY RESOLUTION TEST CHART

AD A092210

SSC ~~Report No.~~ 81980

LEVEL II

# EVALUATION OF THE NASTRAN GENERAL PURPOSE COMPUTER PROGRAM

FINAL REPORT

Submitted to:

Office of Naval Research
Code 474
Arlington, Virginia 22217

by


J. W. Jones
H. H. Fong
D. A. Blehm

Swanson Service Corporation
P. O. Box 5415
Huntington Beach, CA 92646
(714) 964-1552




August 1980

137





8011 19 024

TABLE OF CONTENTS

TABLE OF CONTENTS (cont'd.) Page



A

LIST OF FIGURES

## LIST OF TABLES

## ACKNOWLEDGMENTS

The authors wish to acknowledge the support of this study by the Office of Naval Research, under the sponsorship of Dr. Nicholas Perrone. We would especially like to thank Dr. Robert Nickell for his capable guidance and helpful suggestions. We would also like to express our gratitude to Jerry Harrison, NASA Headquarters, for his help in obtaining a listing of the NASTRAN computer program. In addition, we are grateful to the following individuals who provided data, references, and advice to us during various phases of the study:

Dr. Robert Brugh, COSMIC, Athens, Georgia

Mr. Paul Heller, MARC Analysis Research Corporation, Palo Alto, California

Mr. Myles Hurwitz, Naval Ship Research & Development Center, Bethesda, Maryland

Mr. M. Jeffrey Morgan, Universal Analytics Inc., Playa Del Rey, California

Dr. R. Narayanaswami, MacNeal-Schwendler Corporation, Los Angeles, California

Dr. Al Potvin, Exxon Production Research Company, Houston, Texas

Dr. John Robinson, Robinson and Associates, Dorset, England

Dr. Ed Stanton and Mr. Lou Crain, PDA Engineering, Santa Ana, California

Dr. John Swanson, Swanson Analysis Systems, Inc., Houston, Pennsylvania

# SECTION 1

## INTRODUCTION

NASTRAN is a large, general purpose, finite element computer program used primarily for linear structural analysis. In its ten years of existence, NASTRAN has achieved the distinction of being the program with the most users worldwide, most documentation, most user conferences, most cited, and probably the most expensive to develop.

The four objectives of this study were to:

- Assess the current status (usage, documentation, trends) of public-domain versions of NASTRAN;
- Evaluate its program architecture, and compare it with some other general purpose codes;
- Survey its functional description, and comment on its capabilities/limitations and such items as matrix operations, eigenvalue extraction schemes, element library, material properties, etc; and
- Perform advanced evaluation exercises on some selected structural elements, and test their convergence, completeness, and efficiency.

Obviously, it is impossible in a study of this limited scope to evaluate comprehensively all aspects of NASTRAN. Therefore, at the outset, it was decided to focus attention at the following areas in the first half of the year-long study: user survey, documentation review, program architecture, functional description, and selected verification exercises. The second half of the study concentrated on the advanced evaluation exercises, designed to assess element quality and limitations (see Section 7).

There are literally thousands of NASTRAN users. For convenience in this study, we shall divide them into four categories:

| CATEGORY | DESCRIPTION |
|---|---|
| 1. Expert | This person knows NASTRAN inside out. He is familiar with all the documentation, including the NASTRAN Programmer's Manual. He is adroit with the Direct Matrix Abstraction Program (DMAP). He has used NASTRAN in a variety of problems for probably close to ten years, and is recognized in his organization as the NASTRAN expert, the person to whom everyone comes for help. |

| CATEGORY | DESCRIPTION |
|---|---|
| 2. Experienced | This person knows NASTRAN well, and has probably used the code for five to seven years. He is competent in finite element theory and modeling, and most likely, has used several other codes. He knows how to plot models to his liking with minimum effort. He has used several of the NASTRAN rigid formats and DMAP, and knows the input/output well. This person is usually a senior engineer in a large company. |
| 3. Common User | The common NASTRAN user has generally used the code for two to five years and is familiar with rudimentary finite element theory. He is likely to have used one rigid format (e.g. statics) almost exclusively, to analyze different designs of one product. This person finds the concept of DMAP very difficult. |
| 4. Novice | The new NASTRAN user has zero to two years experience. He is becoming familiar with NASTRAN modeling, input, and output. He needs guidance in modeling techniques, works for the experienced or common user, and is apt to make mistakes in element connectivity and boundary conditions. |

Throughout this report, certain comments and conclusions shall be addressed to each category of users. Most NASTRAN users fall in Categories 2 and 3.

Unlike most current general purpose finite element computer programs developed by individuals, companies, and universities, NASTRAN was originally conceived by and developed under NASA's sponsorship in 1964-1969. After consulting with an ad-hoc committee of finite element experts from the aerospace industry, NASA established a NASTRAN project management office which: administered the software development contracts, released and updated periodically the voluminous documentation, established a systematic error reporting and correction procedure, and shaped the contents of the code. This responsibility was shifted in 1979 to the Computer Software Management and Information Center (COSMIC) at the University of Georgia. NASA was therefore the "code developer" of NASTRAN. This short explanation on NASTRAN development history serves to point out the fact that, unlike other competing codes, COSMIC/NASTRAN currently does not offer a "hot-line service" to answer promptly user questions on the telephone (see end of Section 2).

The future success and usefulness of NASTRAN (as well as any other structural analysis software) is intimately related to the current and future developments in computers. For some excellent recent reviews and projections on this subject,

the reader is referred to References 1 to 6. These articles review computer developments in the past thirty years and project trends into the 1980's. Topics which are covered, most of which will have an effect on the future of NASTRAN, are: operating systems, high-level languages, time sharing, CAD/CAM, minicomputers, interactive terminals, new memory chips, virtual and bubble memories, disc files, distributed processing, and supercomputers ("number crunchers"). The computer industry is on the threshold of very large scale integration (VLSI) chip technology and bubble memories, both of which appeared on a commercial scale in 1979. The future impact of these developments on structural analysis software is difficult to predict. In view of the current proliferation of minicomputers on the structural analysis scene, it was decided at the end of the NASTRAN advanced evaluation (Section 7) to comment briefly on minicomputer versions of NASTRAN and also on pre- and post-processors now commercially available for NASTRAN. The real worth of a code lies in its maintenance and support.

# SECTION 2

## USER SURVEY RESULTS

This section summarizes the findings of a telephone survey of approximately 25 NASTRAN users, most of whom fall in Categories 2 and 3 and work at large aerospace companies. This survey was by no means a comprehensive picture of NASTRAN usage; rather, it was basically a regional consensus which reflected each user's own opinions and experiences. The aims of the survey were to obtain a quick feel of current NASTRAN usage, obtain user comments and experiences, and solicit suggestions for improvement.

The main problem with the survey was to find current users of public-domain versions of NASTRAN. In fact, only a few users of the latest release (Level 17.5) of COSMIC/NASTRAN were found. Many users had learned NASTRAN using the COSMIC version, but had since switched to commercial versions (such as Universal Analytics Inc.'s UAI/NASTRAN and MacNeal-Schwendler Corporation's MSC/NASTRAN). In the past, many government agencies in awarding contracts had dictated that the vendor use NASTRAN for structural analysis, without specifying the particular public or commercial version. Therefore, the vendor could choose the one he preferred. The overwhelming reason for switching to commercial versions like UAI/NASTRAN and MSC/NASTRAN was better user support for day-to-day problems. Other important reasons were significant cost improvements and more efficient solution algorithms.

The following statements summarize some major findings of the survey. Several interesting comments are amplified later.

- There still exists a large group of goverment and industry users who prefer the public-domain versions of NASTRAN, and who like to modify the code to suit their own needs.
- The biggest single complaint of COSMIC/NASTRAN users is the lack of a hot-line service for prompt support. (See comment at the end of this section).
- NASTRAN users generally think of the code favorably, and consider their NASTRAN experience as a valuable asset in their career development.
- Most users confine their analyses to rigid formats and perform linear elastic analysis. A few of the sophisticated users create their own DMAP sequences, but this appears to be quite rare.
- Another big complaint is cost. NASTRAN is rated too costly for small and medium-sized jobs. Efficiency and run-time improvements are reported in several sources (for example, References 14 and 40) comparing MSC/NASTRAN and UAI/NASTRAN to COSMIC/NASTRAN, but the present study will not evaluate this aspect.

- .NASTRAN documentation is bulky, and difficult to read and use (see Section 3).
- .There is a need for an internal mesh generation capability.
- .Many users wished NASTRAN had nonlinear material and geometry capabilities. (Its present nonlinear capability is limited, and consequently will not be evaluated in this study).

Most users agreed that the executive system-functional modules approach in NASTRAN program architecture makes for a very flexible and powerful program (see Section 4). They use a few rigid formats heavily, and rarely have need for creating DMAP sequences. Usually, DMAP alters are used for output variations. The program can be used for matrix manipulations, again a feature due to DMAP and the modular concept. This feature brought out another interesting observation on the program architecture. NASTRAN was conceived and developed ten to fifteen years ago, when computer core was expensive and the modules were created to limit the dependence on core size. Today, core is plentiful and cheap (and rapidly becoming cheaper), virtual memories are commonplace and bubble memories are emerging. Most contemporary programs are manipulated by the machine operating system rather than an executive system internal to the program (such as NASTRAN). This NASTRAN program architecture prevents the user from communicating interactively with the program.

In the area of error and bug correction, the user's views depend largely on the size of his organization and whether there exists a competent in-house staff to correct bugs and implement new elements or modules. The small or medium-sized user finds it virtually impossible to get things corrected on his own on COSMIC/NASTRAN, and will very likely opt for a commercial version. The ger user has the computing and manpower resources necessary for corrections and implementations, and may prefer COSMIC/NASTRAN for its flexibility. Apparently, the released versions of the program from COSMIC have not been thoroughly tested and verified, causing many reported system-type problems and errors. When a user discovers a bug, his only course of action (other than correcting it himself) is to report it to the current maintenance contractor. Then, the government decides on the priority of correcting various reported errors, and the maintenance contractor may not be able to correct the error until the next released level. Another problem is the development effort required for the different machine versions of a given NASTRAN release level. A level is usually developed on one machine, and the testing reliability and quality assurance may not be as high on another machine.

Users complained about possible errors in the capabilities dealing with: substructuring, cyclic symmetry, differential stiffness with thermal loading, axisymmetrical analysis, piecewise linear analysis, eigenvalue extraction methods and their accuracy in computing rigid-body modes for large problems, and the QUAD2 element. Capabilities and options not presently in NASTRAN but desired by many users are: incompressible elements, composite materials, strain recovery, mesh generation, timing estimates, non-zero initial conditions for modal analysis, alternate time integration schemes, and nonlinear geometric and material analyses.

Commentary

A very interesting comment on NASTRAN maintenance was made by several users. COSMIC/NASTRAN was first released to the public in 1969, and has been updated through continuous government contracts. Each new updating contract is bid on by private companies. Then, the successful bidder's responsibility is to define, initiate, integrate, and quality-assure all work done on the program. Therefore, each new level released to the public is a direct reflection of the performance of the maintenance contractor and subcontractors. In the past, continuity and coordination of efforts on some occasions could have been better.

Currently, user support on COSMIC/NASTRAN is coordinated by Dr. Robert L. Brugh, NASTRAN Project Manager at COSMIC, University of Georgia at Athens, Telephone: (404) 542-3265. He is responsible for providing answers to user's questions or problems, and coordinates closely with Computer Sciences Corporation, the current COSMIC/NASTRAN maintenance contractor. COSMIC maintains and updates the NASTRAN Software Problem Report (SPR) log, which can be requested by any user. Several recent reports from COSMIC/NASTRAN users with specific questions and problems (and who decided to use this channel of communication with Dr. Brugh and COSMIC) have been favorable. COSMIC/NASTRAN users are therefore urged to contact COSMIC promptly when they discover system problems, element anomalies, and other bugs.

# SECTION 3

## NASTRAN DOCUMENTATION

Some general comments are made here about NASTRAN documentation, which primarily consists of:

| | |
|---|---|
| .The NASTRAN Theoretical Manual | (Ref. 7) |
| .The NASTRAN User's Manual | (Ref. 8) |
| .The NASTRAN Programmer's Manual | (Ref. 9) |
| .The NASTRAN Demonstration Manual | (Ref. 10) |
| .NASTRAN User's Guide (Level 17.5) | (Ref. 11) |

The first four manuals were designed for reference purposes, and are bulky and sometimes difficult to use. In general, the documentation in these four manuals is fairly complete though of uneven quality. Of the four user categories, the expert user is probably the only one to have examined all these thick volumes. It is often difficult to find information. Examples of this difficulty are given below:

| MANUAL | "DIFFICULT-TO-FIND" INFORMATION (OR MISSING) |
|---|---|
| Theoretical | Number of nodes and degrees of freedom for each element. |
| Theoretical | User hints and limitations of each element. |
| Theoretical | Section 5.8 (80 pages long) adequate description of 15 membrane and bending 2-D surface elements, but element names are missing in half of the subsections. |
| User's | Element summary table showing: classification, name, brief description, nodes, degrees of freedom, material options, and element developer and date. |
| User's | Time and cost estimates for different rigid formats and elements. |
| User's | Tables showing: geometry cards; heat transfer, fluid, mass, rigid and dummy elements; material properties; constraints and partitioning; loads (static, dynamic, heat transfer); problem control. |
| Programmer's | Number of overlay levels and system flow charts for the IBM, UNIVAC, and CDC computer systems. |
| Demonstration | Simple problems explaining input stream line-by-line to the novice user. |

Most experienced and common users have undoubtedly read only those sections in the manuals necessary for their day-to-day use. The last two sections in the User's Manual are particularly helpful and commendable (since they do not usually appear in most other code manuals): the well-catalogued diagnostic messages and the dictionary of NASTRAN terms. An attempt to improve many of the above-mentioned difficulties and missing items in the four manuals of NASTRAN documentation is made in the NASTRAN User's Guide (Level 17.5).

The User's Guide is indispensable to all users of COSMIC/NASTRAN. It gives a concise overview of: the documentation; NASTRAN modeling; descriptions of the various rigid formats; program architecture; input deck description; executive control, case control, and substructure control decks; printing and plotting features; NASTRAN implementation and input decks for the IBM 360/370 (OS), UNIVAC 1108/1110 (EXEC 8), CDC 6000/CYBER (NOS and NOS/BE) computer systems; NASTRAN information sources; estimate of resource and time requirements; DMAP; and application examples for all major types of analyses. This 618-page guide is an admirable attempt to place in one document most of the useful information a user will ever need. The following items are especially valuable: a table summarizing NASTRAN finite element names versus their characteristics (nodes, mass, load types allowed, heat transfer capability, etc.); a general discussion of general modeling restrictions in NASTRAN; a summary table of analysis options for dynamic rigid formats; a comprehensive summary table of modeling options versus each of the 20 rigid formats (15 displacement, 3 heat transfer, 2 aero-elasticity); many examples of executive and case control deck setups; a 13-page summary table of bulk data options (badly needed in front of the User's Manual); a clear explanation of the NASTRAN overlay system; practical guidelines to estimate cost, rigid format resource requirements, CPU time, and other hardware requirements; and finally, many detailed line-by-line input deck explanations of application examples illustrating the proper use of all major rigid-format analyses. Therefore, the User's Guide is strongly recommended as the first NASTRAN document to read for the novice user, and as a handy reference for the common, experienced, and expert users alike.

Other NASTRAN-related documentation which may be of interest:

- NASTRAN User's Experiences and Colloquia (First through Eighth, 1971-1979, available from National Technical Information Service, Springfield, Virginia 22161).

.NASTRAN Software Problem Report (SPR) Log (periodically updated and published by COSMIC).

.NASTRAN Newsletter (free subscription, from COSMIC).

.Documentation Error Reports (DER) (COSMIC).

.NASTRAN - NASA Structural Analysis (1979) (COSMIC).

.MSC/NASTRAN Basic Training Manual (Ref. 12).

.MSC/NASTRAN Primer: Static and Normal Modes Analysis (Ref. 13).

.A Brief Description of MSC/NASTRAN (Ref. 14).
(compares MSC/NASTRAN and Level 16 COSMIC/NASTRAN, as of April, 1978;
This document is scheduled to be updated in Spring 1981 and will compare MSC/NASTRAN to Level 17.5 Cosmic NASTRAN

.MSC/NASTRAN Applications Manual (Ref. 53)

.MSC/NASTRAN Demonstration Manual

.MSC/NASTRAN Handbook for Linear/Static Analysis (June 1980)

# SECTION 4
# PROGRAM ARCHITECTURE

The key concept behind NASTRAN program architecture is modularity. When NASTRAN was originally designed, certain design criteria were stipulated (References 7, 9, 11), such as: simplicity of input, minimization of chances for human error and the need for human intervention during program execution; functional independence of solution modules; ease of program modification and extension; versatility and adaptability for various computer systems; restart capability, etc. The resulting NASTRAN program architecture is shown in Figure 4-1, which illustrates the major functional subdivisions and their interrelationships.

The two major components are the Executive System and the Functional Modules. The Executive System is the heart of the system. It executes the program in two phases: the preface (where a basic setup is performed); and the execution of the DMAP program in which the modules are controlled. The preface does extensive pre-processing and sorting of bulk data (this function of NASTRAN is superior to most general purpose codes), generates initial file allocation tables, and initializes the problem. The Executive System establishes, protects, and communicates values of parameters for each module. It allocates system files to all data blocks generated during program execution; a file is "allocated" to a data block, and a data block is "assigned" to a file. It also maintains a full restart capability for restoring program execution after either a scheduled or unscheduled interruption. The general philosophy in programming Executive System routines is that reliability and efficiency are paramount concerns. Few changes were anticipated in the Executive System as NASTRAN grew. Therefore, a few general rules were imposed on the Executive System code, resulting in a sophisticated, but difficult to modify, system (Ref. 11). The Executive System is dependent on a particular computer and its operating system. It comprises approximately 10 percent of the total code.

The Functional Modules are self-contained subprograms which may not call, or be called by other modules. They may be entered only from the Executive System. Each module has: data blocks (an important type is a matrix), subroutines (which can communicate with each other within a module); parameters; and drivers. A change in one module affects no others, as long as the interface with the Executive System is preserved. No module can directly specify or allocate physical files.

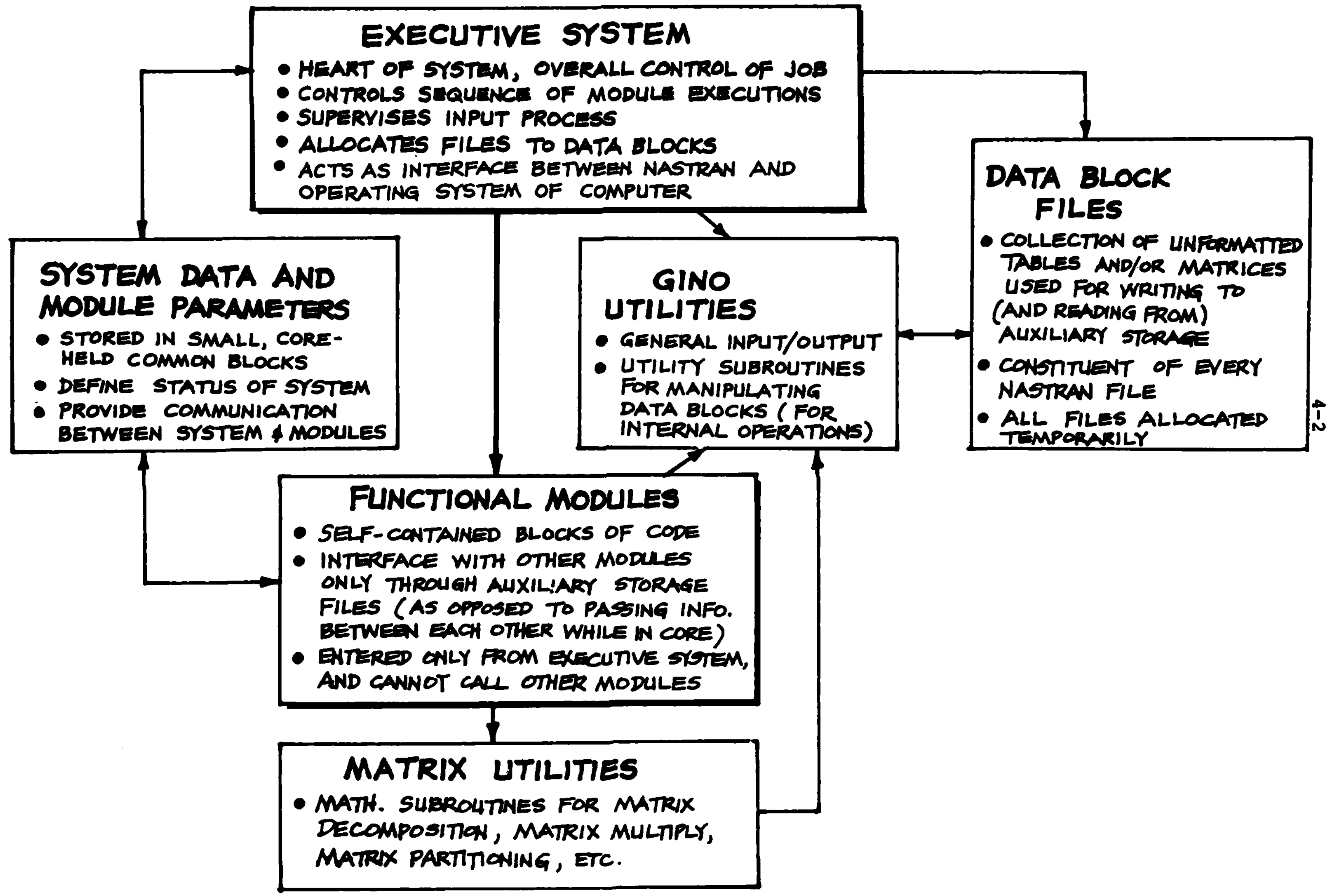



FIGURE 4-1 NASTRAN PROGRAM ARCHITECTURE AND FUNCTIONAL SUBDIVISIONS

A module can vary in size from less than ten lines of code to many thousands of executable statements. Four types of modules exist in NASTRAN, each with its specific range of functions: preface modules; executive operation modules; utility modules; and analysis-oriented modules (Ref. 11). Communication between modules is performed using data block files and parameters (stored in the blank common block). Functional modules are machine-independent, and comprise about 90 percent of the code.

Input/output is supported through GINO (Generalized Input/Output) , a comprehensive set of machine-dependent utility subroutines developed for manipulating data blocks in internal NASTRAN operations. The basic unit of I/O is a "logical record", whose word length is variable. Each column of a NASTRAN matrix data block is one logical record. Two subroutines convert special NASTRAN input card formats to standard FORTRAN data words easily handled by all NASTRAN input processors. All NASTRAN routines must use GINO. They are consequently isolated from the actual physical hardware and such concerns as blocking factors and device characteristics. Since main memory is used as a scratch pad (no module may leave values in main memory), GINO-formulated data blocks form the bulk of the intermodule communication (Ref. 14).

NASTRAN is primarily a file-oriented system, using mass storage devices for nearly all major data transfers. Four types of input/output systems are provided in NASTRAN (Ref. 11): external user-supplied interfaces; internal data blocks (used for temporary data storage); checkpoint/restart files; and substructure operating files (storing all data necessary for a complete multi-stage substructuring analysis). NASTRAN has a sophisticated checkpoint and restart capability, offering the user four general types of restarts: unmodified restart, pseudo-modified restart, modified restart and rigid formal switch. The reader is referred to References 7, 8, and 11 for details of restarting.

NASTRAN is not a core program. It is designed to dynamically allocate core so that at execution time, all the core memory that can be made available will be used. "Open core" memory management means: a contiguous block of randomly addressable working storage defined by a labeled common block, whose length is a variable determined by the NASTRAN executive subroutine KORSZ.

Main memory is treated as a large single-dimensional scratch array by all modules. The length is communicated dynamically to the module by the Executive System. Fixed-dimension statements for arrays are not allowed in any module as all user input is open-ended (Ref. 14). Spill logic is provided to transfer data to scratch files if complete core allocation is impossible. NASTRAN was not originally designed to operate under a time-sharing system.

Each module in NASTRAN is assigned separate overlay segments independent of the other modules. During execution, only the module code and the required utility subroutines are loaded into core. Furthermore, a common block assigned by the module is loaded below the current code for use in an "open-core" storage space. Open core exists from this point to the maximum core space requested for the execution. All matrix and table data required for a given module operation are brought into open core for temporary storage. Results are moved from open core to data block files for access by other modules. A typical NASTRAN overlay tree is illustrated in Figure 4-2. Examples of modules are shown; these are divided into separate parts, consisting of a small "driver" routine which calls the major sections of code located near the bottom of core. This scheme allows the large and complex matrix utility routines to be used by more than one module in a link, yet still allows the largest possible open core space for each module (Ref. 11).

COSMIC/NASTRAN and the open core concept are implemented on three major computer systems: IBM 360/370 series, UNIVAC 1108/1110 series, and CDC 6600/CYBER series. The NASTRAN program is divided into a series of logical pieces called "links". Each link is a complete program or load module, and contains its own overlay structure and its own root segment (the set of subprograms which is always resident in main memory for that link). Communication between links occurs through computer files. Table 4-1 illustrates some characteristics of NASTRAN implementation on the three computer systems.

Table 4-2 compares some program architecture features of NASTRAN with four general purpose codes: MARC, ADINA, AGGIE, and ANSYS. The only code among these give to offer a general matrix manipulation capability is NASTRAN.

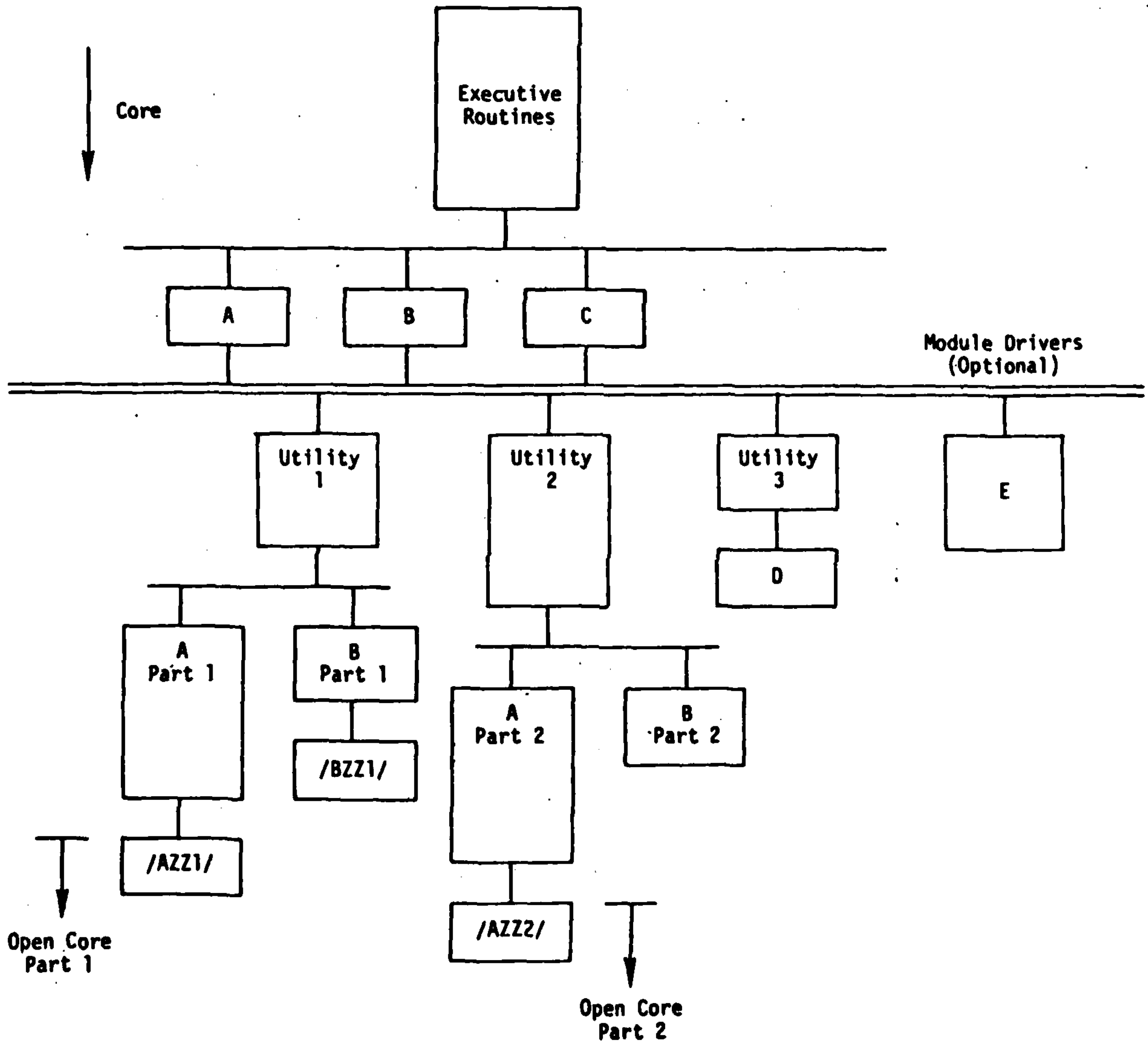



FIGURE 4-2. ILLUSTRATION OF NASTRAN OVERLAY SYSTEM.

Two recent articles which survey and compare general purpose code capabilities and features are Fredriksson and Mackerle (Ref. 21) and Dunder and Belonogoff (Ref. 22).

TABLE 4-1

NASTRAN IMPLEMENTATION ON 3 MAJOR COMPUTER SYSTEMS

| | IBM 360/370 Series | UNIVAC 1108/1110 Series (Exec 8 OS) | CDC CYBER/6000 Series (NOS) |
|---|---|---|---|
| Word size | 32 bits | 36 bits | 60 bits (single precision) |
| Character capacity | 8 bits/character<br>4 characters/ word | 6 bits/character<br>6 characters/ word | 6 bits/character<br>10 characters/word |
| Number of links | 16 | 15 | 15 |
| Overlay | Open core, labeled common block | Open core segment loader, common block | Segmentation loader, segments loaded dynamically |
| Input/Output | GINO consists of 3 decks:<br>.GINO<br>.NASTIO<br>.OPEN | 3 categories of I/O routines:<br>.Identify and obtain logical files GNFIAT<br>.Perform actual I/O (GINO, IO1108)<br>.SGINO: generate plot tape | 4 ways:<br>.Card input, printed/ punched output, binary output<br>.All other I/O uses GINO<br>.SGINO: Plot output<br>.SOFIO: substructuring I/O |

TABLE 4-2

COMPARISON OF PROGRAM ARCHITECTURE OF FIVE GENERAL PURPOSE CODES

| FEATURE | NASTRAN (Refs. 7,9,11) | MARC (Ref. 15) | ADINA (Refs. 16,35) | AGGIE I (Ref.19) | ANSYS (Ref.20) |
|---|---|---|---|---|---|
| 1. Root Segment in Overlay Structure | IBM:NASTRAN UNIVAC: MAIN1→ MAIN15 CDC:NAST01→ NAST15 | CONTRO | AAMAIN | MAIN (plus 9 other subroutines in control phase) | None |
| 2. Data Input | GINO (machine-dependent subroutine) | OAREAD (first level overlay) | ADINI | NONSPI (most important of 20 subroutines in input block) | 1 subroutine processes element and *nodal info.* |
| 3. No. of overlay levels | 3 to 7 (depends on system) | 5 (mostly 3) | 4 (same as number of execution phases) | 4 | 3 to 6 (depends on system) |
| 4. Solver option .In-core .Out-of-core | No<br>Yes | Yes (subroutine)<br>Yes (random access; sequential) | No<br>Yes (COLSOL) | Yes (OPTSOL)<br>Yes (OPTBLK) | Yes<br>Yes |
| 5. Dynamic Storage Allocation | Yes, labeled common block | Yes, labeled common block | Yes, blank common block with variable length | Yes, labeled common block | Extended core capability, random access files |
| 6. Matrix storage scheme | PACK/UNPACK by columns (PAKUNPK) | by columns | "Skyline" scheme (by columns) | "Skyline" scheme (by columns) | Wave-front scheme |
| 7. Interactive Mesh Generation | No | Yes | No | No | Yes |
| 8. General matrix operations capability | Yes (DMAP) | No | No | No | No |

# SECTION 5

## FUNCTIONAL DESCRIPTION

This section briefly highlights some NASTRAN functional descriptions, in order to give the reader an overview of the versatility and capabilities of the code. These descriptions include:

- .A rigid format - What is it? How many are there? and a typical DMAP sequence for linear statics analysis
- .Direct Matrix Abstraction Program - What is DMAP? and a short example
- .Structural Elements - a summary table giving element categories, names, descriptions, number of nodes and degrees of freedom at each node, material property, and origin
- .Constitutive library - material property options
- .Constraints and partitioning
- .Matrix operations
- .Eigenvalue extraction methods } compared to other general purpose codes
- .Time integration scheme } compared to other general purpose codes
- .Plotting capabilities
- .Restart capability
- .Nonlinear capabilities
- .Special NASTRAN features - not usually found in other codes
- .Substructuring - automated multi-stage substructuring analysis

### 5.1. *Rigid Formats*

A total of 20 rigid formats exists in Level 17.5 of COSMIC/NASTRAN. A rigid format is an established sequence of DMAP instructions stored in the Executive System to perform some standardized analyses, such as:

- .Static analysis - conventional linear static analysis; with inertia relief (inertia effects of unconstrained rigid body accelerations); with differential stiffness (nonlinear effects of large deflections)
- .Normal modes analysis (linear elastic models)
- .Buckling analysis
- .Piecewise linear (material plasticity) analysis
- .Direct complex eigenvalue analysis
- .Direct frequency and random response
- .Direct transient analysis
- .Modal complex eigenvalue analysis
- .Modal frequency and random response
- .Modal transient analysis
- .Normal modes with differential stiffness
- .Static analysis using cyclic symmetry
- .Normal modes analysis using cyclic symmetry
- .Heat transfer analysis - linear or nonlinear steady-state; transient
- .Aeroelasticity analysis - modal aerodynamic flutter analysis; modal aeroelastic response due to applied loads and gusts

Rigid formats are identified with an integer number and an analysis approach. There are three approaches: 15 rigid formats in displacement (APProach DISP); 3 in heat transfer (APProach HEAT); and 2 in aerodynamic (APProach AERO). Page 4.12-3 of Reference 11 contains an excellent table which summarizes the various modeling options offered versus rigid format number. These rigid formats provide the typical user with a tremendous range of analysis options, using pre-established module call sequences (the usual case for all but the expert user). If he chooses, the expert/experienced user can modify the rigid formats by DMAP alters for a particular problem.

Table 5-1 shows the DMAP sequence for a typical rigid format, in this case the flow sequence of the 30 major functional modules for linear statics analysis. Note that the modules have been classified into pre-processing, analysis, and post-processing modules. This is the only rigid format with a fully-stressed design optimization capability, and the optimization loop is shown. SSG3 is the key analysis module here since it solves for the independent displacements. The rigid format for normal modes analysis has a very similar DMAP sequence to that of linear statics. The key analysis module for normal modes analysis is named READ (real eigenvalue analysis - displacement), consisting of some 52 subroutines.

## 5.2. Direct Matrix Abstraction Program (DMAP)

DMAP is one of the most powerful matrix manipulation tools offered in any code. It is the programming language of NASTRAN. DMAP is simply a language of macro instructions which enables the user to sequence any combination of matrix operations. A DMAP instruction has the form:

```
(Module Name)     (Input Data Blocks)/(Output Data Blocks)/(Parameter List)$
```

where the slash is used as a delimiter between data blocks and the dollar sign is used to terminate the DMAP instruction.

A short DMAP example is explained below (Ref. 14):

To compute $[C] = [A] + [B]$

$[D] = [A]\,[C]$

The DMAP sequence is:

```
BEGIN $ START DMAP
ADD  A,B/C/$   ADDS A TO B
```

TABLE 5-1. DMAP SEQUENCE FOR RIGID FORMAT 1 - LINEAR STATICS

| | | Module Name | FUNCTION |
|---|---|---|---|
| | Pre-processing Modules | GP1 | Coordinate system, grid point locations, relate internal to external grid points |
| | | GP2 | Element connection table |
| | | PLOT | Undeformed structure plots |
| | | GP3 | Static loads, grid point temperatures |
| | | TA1 | Element tables for use in matrix assembly and stress recovery |
| | | OPTPR1 | Phase 1 property optimization, initialization check |
| OPTIMIZATION LOOP | Analysis Modules | EMG | Element stiffness and mass matrices, for later assembly |
| | | EMA | Element matrix assembler assembles stiffness matrix, grid-point table singularity |
| | | GPWG | Weight and balance information |
| | | SMA3 | Adds general elements to stiffness matrix |
| | DMAP Loop | GP4 | Displacement sets, MPC equations, enforced displacement vector |
| | | GPSP | Determines if possible grid point singularities remain |
| | | MCE1 | Positions MPC equations |
| | | MCE2 | Partitions stiffness matrix |
| | | SCE1 | Partitions out SPCs |
| | | SMP1 | Partitions constrained stiffness matrix |
| | | RBMG1 | Partitions out free-body supports |
| | | RBMG2 | Decomposes constrained stiffness matrix |
| | | RBMG3 | Rigid body transformation and check matrices, error ratio |
| | | SSG1 | Generates static load vectors |
| | | SSG2 | Applies constraints to static load vectors, calculates determinate reactions |
| | | SSG3 | Solves for independent displacements |
| | | SDR1 | Recovers dependent displacements and SPC forces |
| | | GPFDR | Calculates grid point force balance element strain energy as requested |
| | | SDR2 | Calculates element forces and stresses |
| | Post-processing Modules | SDR3 | Prepares requested output sorted by grid point number of element number |
| | | XYTRAN | Prepares input for X-Y plots |
| | | XYPLOT | Prepares X-Y plots of displacements, stresses, forces, SPC forces vs. subcase |
| | | OPTPR2 | Performs Phase 2 property optimization |
| | | PLOT | Deformed structure and contour plots |

```
MATPRN C//$        PRINT C
MPYAD A,C,/D/$     COMPUTE D
MATPRN D//$        PRINT D
END $  TERMINATE
```

This simple DMAP example provides an illustration of the use of DMAP which is easy to understand even for the relatively inexperienced novice and common users of NASTRAN. In addition to matrix arithmetic, DMAP in NASTRAN can be used for tasks categorized as: executive, utility, and structural (e.g., assemble tables, calculate element data).

A rigid format sequence may be modified by the user using the ALTER capability. A library of useful pre-defined DMAP alter sequences exists, and these sequences are called RFALTERS. The user must merge RFALTERS into the Executive Control Deck by using a machine-dependent operating system utility for merging files.

### 5.3. Structural Elements

Table 5.2 gives a summary of the structural elements available in Level 17.5 of COSMIC/NASTRAN. Of the 33 elements listed, 4 are higher-order elements (TRIM6, TRPLT1, TRSHL, TORDRG) and 4 are isoparametric elements (QDMEM1, IHEX1, IHEX2, IHEX3). Nineteen elements have heat transfer capability. For simplicity, fluid elements and some scalar elements have not been included in Table 5-2, such as: viscous damper, scalar spring and mass, concentrated mass, general element, and dummy element. By today's standards, the element library in NASTRAN is rated only fair. Many of the elements are about 10 to 15 years old and have existed in NASTRAN since its inception. One glaring weakness is the absence of a quadrilateral solid-of-revolution element for plane strain, plane stress, or axisymmetric analyses. The trapezoidal ring element TRAPRG, which requires two sides of the element to be parallel, is simply too restrictive and not versatile enough. A major portion of this study was to perform advanced evaluation exercises on selected NASTRAN elements. The elements selected for advanced evaluation are noted in Table 5-2, and the reader is referred to the appropriate subsection in Section 7 for evaluation results. Page 3.2-2 of Reference 11 contains a detailed summary table of NASTRAN finite element characteristics (e.g. thermal loads, lumped or consistent mass, differential stiffness, plasticity, heat transfer, etc.). A library of more than 60 different finite element formulations is offered in COSMIC/NASTRAN Level 17.5.

TABLE 5-2. STRUCTURAL ELEMENTS SUMMARY (1 of 2)

| Category | Name | Advanced Evaluation Subsection | Element Description | Nodes | DOF's at ea. Node | Material Property* | Remarks |
|---|---|---|---|---|---|---|---|
| Linear (1-D) | ROD | | Tension, torsion only | 2 | 2 | I | |
| | CONROD | | Tension, torsion only, with property | 2 | 2 | I | Undocumented in Theoretical manual |
| | TUBE | | Tension, torsion only | 2 | 2 | I | |
| | BAR | | Tension, torsion, bending, and shear | 2 | 6 | I | MSC/NASTRAN also has BEAM & BEND elements |
| Surfaces (2-D) | SHEAR+ | | In-plane shear panel, quadrilateral | 4 | 1 | I | No thermal expansion |
| | TWIST+ | | Twist panel | 4 | 1 | I | Not in MSC/NASTRAN |
| | TRMEM+ | | Triangular membrane (no bending) | 3 | 2 | I,A | Constant strain triangle |
| | TRIM6 | 7.4 | Linear strain membrane triangle | 6 | 2 | I,A | Argyris (1965), Cowper et. al. (1968), Zienkiewicz (1971) |
| | QDMEM+ | 7.4 | Quadrilateral membrane (no bending) | 4 | 2 | I,A | 4 overlapping TRMEM's |
| | QDMEM1+ | 7.4 | Isoparametric form of QDMEM | 4 | 2 | I,A | Taig-Irons (1966) |
| | QDMEM2+ | | Quadrilateral membrane, 1 center node | 4 | 2 | I,A | 4 non-overlapping constant strain TRMEM's |
| | TRPLT+ | | Bending, transverse shear | 3 | 3 | I,A | Clough triangle, centroid node |
| | TRPLT1+ | 7.4 | Nonconforming, quintic polynomial for transverse displacement | 6 | 3 | I,A | Narayanaswami (1974) Cowper et. al. (1968) |
| | QDPLT+ | 7.2 | Quadrilateral bending, transverse shear | 4 | 3 | I,A | 4 basic TRPLT's |
| | TRBSC+ | | Basic unit for all elements except TRPLT1, triangular bending | 3 | 3 | I,A | Clough-Tocher (1965), not in MSC/NASTRAN |
| | TRIA1+ | | Triangular membrane plus bending, also transverse shear | 3 | 5 | I,A | Sandwich plate |
| | TRIA2 | | Triangular membrane plus bending, solid cross-section | 3 | 5 | I,A | MSC/NASTRAN also has TRIA3 and TRIA6 |
| | QUAD1+ | | Quadrilateral membrane and bending | 4 | 5 | I,A | Similar to TRIA1, QDMEM + QDPLT |
| | QUAD2+ | 7.3 | Quadrilateral like TRIA2 | 4 | 5 | I,A | MSC/NASTRAN has isoparametric QUAD4 and QUAD8; UAI/NASTRAN has isoparametric QUAD3 |

TABLE 5-2. STRUCTURAL ELEMENTS SUMMARY (2 of 2)

| Category | Name | Advanced Evaluation Subsection | Element Description | Nodes | DOF's at ea. Node | Material Property* | Remarks |
|---|---|---|---|---|---|---|---|
| Shells | TRSHL+ | 7.3 | Higher-order thin shell element, quadratic membrane displacements, quintic normal displacement, non-conforming | 6 | 5 | I | Sum of TRIM6 and TRPLT1, Narayanaswami (1974), Novozhilov shallow shell theory |
| | CONEAX[a] | 7.5 | Conical shell element, with transverse shear capability, axisymmetric | 2 | 5 | I | Cannot combine with other elements; problems with thermal loads (?) |
| | TORDRG+ | | Toroidal shell element and shell cap, cubic membrane displacement, quintic flexural displacement | 2 | 2 | I,0 | Mallett-Jordan (1969) MAGIC code |
| Solids of Revolution | TRIARG[a] | | Triangular ring, axisymmetric loading | 3 | 3 | I,0 | Mallet-Jordan's MAGIC (1969) and MAGIC II (1971) codes, based on Clough-Rashid (1965) and Wilson (1965) papers |
| | TRAPRG[a] | 7.6 | Trapezoidal ring, axisymmetric loadings | 4 | 3 | I,0 | |
| | TRIAAX+ | | Triangular ring, nonaxisymmetric loading | 3 | 3 | I,0 | |
| | TRAPAX+ | 7.6 | Trapezoidal ring, nonaxisymmetric loading | 4 | 3 | I,0 | MSC/NASTRAN has linear strain triangular ring TRIAX6 |
| Solids (3-D) | TETRA | | Constant-strain tetrahedron | 4 | 3 | I | |
| | WEDGE+ | | Constant-strain wedge | 6 | 3 | I | MSC/NASTRAN offers PENTA (6-15 nodes) and HEXA (8-20 nodes) |
| | HEXA1+ | 7.7 | Constant-strain brick, 5 tetrahedra | 8 | 3 | I | |
| | HEXA2+ | 7.7 | Constant-strain brick, 10 overlapping tetrahedra | 8 | 3 | I | |
| | IHEX1+ | 7.7 | 8-node isoparametric solid, linear displacement variation | 8 | 3 | I | Suitable for problems with large shear stresses |
| | IHEX2+ | 7.7 | 20-node isoparametric solid, quadratic displacement variation | 20 | 3 | I | IHEXi elements based on theoretical work by: Irons & Zienkiewicz (1966-1968), Clough (1969), Zienkiewicz, Taylor, Too (1971), Pawsey-Clough (1971) |
| | IHEX3+ | | 32-node isoparametric solid, cubic displacement variation | 32 | 3 | I | |

+Not in library of Version 60 of MSC/NASTRAN (May 1980)

*I = isotropic, O = orthotropic, A = anisotropic

[a]Not recommended for use in Version 60 of MSC/NASTRAN (May 1980)

### 5.4. Constitutive Library

As already indicated in Table 5-2, NASTRAN allows three basic types of material properties: isotropic; orthotropic (for TORDRG and solid-of-revolution elements only); and anisotropic (2-D flat surface elements only). In addition to these elastic properties, NASTRAN material data cards are also used to define mass density, thermal expansion coefficients, and stress limits. Each material is given an unique identification number which may be referenced by any number of property cards. Material data cards are identified in the Bulk Data Deck as follows:

.Temperature-independent material properties (MATi)

| | |
|---|---|
| MAT1 | elastic isotropic |
| MAT2 | anisotropic, 2-D flat surface elements only |
| MAT3 | elastic orthotropic, TORDRG and solid-of-revolution elements only |
| MAT4 | heat transfer analysis, isotropic properties |
| MAT5 | heat transfer analysis, anisotropic properties |

.Temperature-dependent material properties

| | |
|---|---|
| MATTi | temperature dependence in conjunction with MATi |
| TABLEMi | tabular functions of temperature-dependent properties |

.Stress-dependent material properties

| | |
|---|---|
| MATSi | stress dependence in conjunction with MATi |

### 5.5. Constraints and Partitioning

NASTRAN offers the user a variety of options to constrain and reduce his problem. These constraints are used to specify fixed boundary conditions, to eliminate matrix singularities, to define rigid elements, and to support free body motion. The most important of these are: multiple point constraints (MPC's), single point constraints (SPC's), OMIT and ASET. A multipoint constraint is a linear relationship between two or more displacement degrees of freedom. Closely related to MPC's are the rigid elements (RIGDR, RIGD1-3) in NASTRAN, which are very stiff connections. These can be used to model levers, pulleys, gear trains, and rigid links to remove matrix ill-conditioning. SPC's are vectors of enforced displacements, any or all of whose elements may be zero. SPC's commonly represent structural boundary conditions of zero displacement and slope, or enforced deformations at nodes.

For matrix reduction the user can select either the OMIT card or the ASET card, but not both. OMIT specifies degrees of freedom to be reduced out of

the analysis set, while ASET specified independent degrees of freedom to be retained. Since they represent complementary sets, the user should specify the smaller of the two. The specified degrees of freedom for elimination by reduction are restricted to only nonconstrained coordinates. Another bulk data card, SUPORT, may also be used to specify the degrees of freedom which will remove rigid body motion. SUPORT is used to supply temporary, nonredundant constraints on free body motion. It is required for inertial relief analysis (Rigid Format 2), and recommended for use in modal analysis of free bodies.

Errors in constraint data are not automatically corrected by NASTRAN. Conflicting constraint data, when detected, are treated as fatal errors (Ref. 11). Grid point singularity tests are performed to detect null columns in the stiffness matrix resulting from deficient constraints or element connectivities.

5.6. Matrix Operations

An unique feature of NASTRAN is the user's option to manipulate matrices using DMAP. Therefore, theoretically, he can solve any numerical analysis problem which can be expressed in matrix form. Elements of matrices can be specified by the DMI card. The user can build a DMAP sequence using any combination of 43 matrix handling subroutines. Design philosophy of the modules is strictly based on handling large matrices. All matrices are stored on peripheral devices (tapes, disks and/or drums) by columns, and are packed in non-zero strings. Matrix sparsity and bandedness are emphasized and utilized. The user may select single or double precision control by the executive control card PREC; this is the only method offered in NASTRAN to combat round-off error accumulation. This matrix handling feature in NASTRAN is elaborate, sophisticated, and powerful. (The only other general purpose code which offers a similar matrix operations capability is SPAR.)

5.7. Eigenvalue Extraction and Time Integration

Table 5-3 compares the eigenvalue extraction methods and direct time integration scheme of NASTRAN with eight other U. S. general purpose codes. This comparison reveals that while NASTRAN offers more eigenvalue extraction routines (4) than any other code, its one time integration scheme is less than an average of two schemes offered by the other codes. The most recent method of eigenvalue

extraction, subspace iteration (Ref. 51), is implemented in only three codes: ADINA, SAP6, and EASE2.

For direct time integration, three methods have gradually emerged to be the most popular among U. S. code developers: Newmark Beta, Wilson and Houbolt. Newmark Beta and Wilson are "implicit" methods, where a matrix system is solved, one or more times per step, to advance the solution. Both algorithms are unconditionally stable, but recent research has indicated that the Wilson algorithm introduces damping into the solution, is sensitive to time step size, and may require more steps to obtain the same accuracy as the Newmark method. The Houbolt method is a third-order backward difference scheme which is unconditionally stable for all time step sizes. However, it introduces damping into the solution, in addition to having a tendency to remove higher modes from the system. The central difference scheme offered in ADINA and MARC is an "explicit" method, where the solution may be advanced without storing a matrix or solving a system of equations. It requires very small time steps, but can be cheaper in cost per time step. For nonlinear dynamic problems, the most sophisticated codes are ADINA, MARC, and a new code called ABACUS being developed by Hibbitt and Karlsson, Inc.

Table 5-4 is a detailed comparison of the four NASTRAN eigenvalue extraction methods. The method desired for a particular solution is selected by the user on the EIGR card. The two tridiagonalization methods (Givens and FEER) are categorized as "transformation" methods (obtaining all the eigenvalues at once) and can be used only for real matrices. The newer FEER method is a tridiagonalization procedure based on the Rayleigh-Ritz method. It combines the best features of the inverse power and Givens methods. The other two methods (inverse power and determinant) are "tracking" schemes and can also be applied to complex matrices. Proper selection of the extraction method depends on the number of eigenvalues desired, matrix character, problem size, and accuracy demanded; Table 5-4 offers some hints in this choice. This subject is discussed in more detail in Ref. 51 and Chapter 13 of Ref. 13.

TABLE 5-3. COMPARISON OF EIGENVALUE EXTRACTION AND TIME INTEGRATION METHODS IN TEN U. S. GENERAL PURPOSE CODES

| | CODE | EIGENVALUE EXTRACTION METHODS | TIME INTEGRATION SCHEMES |
|---|---|---|---|
| 1 | NASTRAN (Ref. 7,11) | 1. Tridiagonal (Givens)<br>2. Inverse Power with Shifts<br>3. Determinant Method<br>4. Tridiagonal Reduction Method - or Fast Eigenvalue Extraction Routine (FEER) | Newmark Beta ($\beta = \frac{1}{3}$) method<br>.For heat transfer transient analysis: user's choice $0<\beta<1$ |
| 2 | MARC (Ref. 15) | Inverse power sweep | 1. Central difference<br>2. Newmark Beta ($\beta = \frac{1}{4}$)<br>3. Houbolt method |
| 3 | ADINA (Ref. 17,35) | 1. Determinant Search Method (with Sturm sequence)<br>2. Subspace iteration | 1. Central difference<br>2. Wilson $\Theta = 1.4$<br>3. Newmark Beta |
| 4 | AGGIE I (Ref. 19) | (unnamed) "tracking" scheme | 1. Newmark Beta<br>2. Wilson $\Theta = 1.4$ |
| 5 | ANSYS (Ref. 20) | Jacobi | Houbolt |
| 6 | STARDYNE (Ref. 23) | 1. Householder-QR Method (tridiagonalization)<br>2. Inverse iteration (same as NASTRAN's inverse power with shifts)<br>3. Lanczos modal extraction method (same as NASTRAN's FEER method) | None (planned for implementation, but undocumented) |
| 7 | SAP6 (Ref. 24) | 1. Determinant Search<br>2. Subspace Iteration | Wilson $\Theta = 1.4$ |
| 8 | EAC/EASE2 (Ref. 25) | 1. Determinant Search<br>2. Subspace Iteration | 1. Newmark Beta<br>2. Wilson $\Theta$ |
| 9 | SDRC/SUPERB (Ref. 26) | 1. Determinant Tracking (with Sturm sequence property)<br>2. Jacobi (with Guyan reduction) | None (NASTRAN & ANSYS interface packages offered) |
| 10 | MSC/NASTRAN (Ref. 41) | 1. Modified Givens (MGIV)<br>2. Inverse<br>3. Givens | Newmark Beta Method<br>Analytical solution of equation (Quasi-closed form) (also has component mode synthesis and superelement dynamic capability and generalized dynamic reduction |

TABLE 5-4. COMPARISON OF NASTRAN EIGENVALUE EXTRACTION METHODS

| | Tridiagonal Method (Givens) | Inverse Power with Shifts | Determinant Method | Tridiagonal Reduction Method (FEER) |
|---|---|---|---|---|
| 1. Type of Method | Transformation | Tracking | Tracking | Transformation |
| 2. Most general form of matrix | $[A-pI]$ | $[Mp^2+Bp+K]$ | $[A(p)]$ | $[Mp^2+K]$ |
| 3. Restrictions on Matrix Character | A real, symmetric, constant; nonsingular mass matrix | M,B,K constant | None | M,K real, symmetric, constant ; M may be singular |
| 4. Obtains eigenvalues in order | All at once, in the order of highest to lowest frequency | Nearest to shift point (highly accurate roots regardless of order of appearance in frequency) | Usually nearest to starting point | Vibration: nearest to shift point<br>Buckling: lowest eigenvalue first |
| 5. Takes advantage of bandwidth | No | Yes | Yes | Yes (and of sparsity) |
| 6. No. of calculations, order of (n = no. of equations b = semi-bandwidth E = no. of eigenvalues extracted) | $O(n^3)$ Tridiagonalize first using Givens; then eigenvalues & eigenvectors using Q-R (Francis, Ortega & Kaiser) | $O(nb^2E)$ (Linearly proportional to number of extracted eigenvalues) | $O(nb^2E)$ | $O(n(b+E)^2)$ Single initial shift, 1 matrix decomposition |
| 7. Handles closely-spaced eigenvalues? | No explicit mention; but can handle double eigenvalues | Known to converge slowly, but corrected by shift strategy | Yes, provides for shift from starting point | No explicit mention |

TABLE 5-4 *cont'd.*

| | Tridiagonal Method (Givens) | Inverse Power with Shifts | Determinant Method | Tridiagonal Reduction Method (FEER) |
|---|---|---|---|---|
| 8. Efficiency, and Limitations | Least efficient, most effort expended before extracting first eigenvalue; can't use for buckling and complex roots; good for large bandwidth problems needing all the modes | Most efficient when only a few eigenvalues are required. These 2 methods are better for problems with a narrow bandwidth, with inverse power method slightly more efficient overall. Problems with a very narrow bandwidth (e.g. beam) favors determinant method. Lengthy computer runs required to obtain a large number of modes. User must estimate frequencies and number of modes in region. | | Major efficiency improvement over tridiagonal method. Computation effort is proportional to number of extracted eigenvalues. Probably most efficient for getting several modes of a real eigenvalue problem. Sparse and banded matrices desirable. |
| 9. Adequacy of Documentation | Adequate, but needs simple example. Good description of flow diagram and algorithms | Good. Fair flow charts but coding description missing. | Fair. Muller's quadratic method and Wilkinson's convergence criterion explained | Good, but too long and difficult to understand |
| 10. Special Comments | Efficient only when matrix size is small enough (250) to be held in core. Produces numerical round-off errors for low-frequency modes at or near zero frequency. (MSC/NASTRAN has a modified Givens method) | Good method, expecially for bifurcation buckling loads and modes. Effectiveness sensitive to problem size. Static model requires little modification for a normal modes solution. | Insensitive to form of [A(p)]. Can use for hydroelastic and aeroelastic problems. Much slower than inverse power method. Now eliminated in Version 60 of MSC/NASTRAN (May 1980). | Very efficient core requirements. Useful compromise between the Givens and inverse power methods. |

## 5.8. Plotting Capabilities

The user specified undeformed or deformed structure plots at the end of the Case Control Deck. The structure plot request packet begins with the card OUTPUT (PLOT). The plotting capabilities in NASTRAN are excellent, comprehensive, and well-documented (Ref. 7, 8, 11) with many sample plot card setups. The following types of plots may be selected:

- .Undeformed geometry plots for all rigid formats
- .Static deformations, and stress/displacement contours
- .Modal deformations (eigenvectors or mode shapes)
- .Transient response or frequency response-vectors or deformed shape for specified times or frequencies
- .X-Y graphs of transient/frequency response
- .V-f and V-g graphs for flutter analysis
- .Topological displays of matrices-showing nonzero element locations when the SEEMAT utility module is requested

NASTRAN plotting is compatible with most plotter hardware like Stromberg-Carlson and CALCOMP. The user has a choice of three types of projections: orthographic, perspective and stereoscopic (microfilm plotters only). Plot labeling and scaling are user-controlled or automatically provided. NASTRAN plotting does not have hidden-line capability, which exists in codes such as MARC and ANSYS. Currently, no interactive pre-processing and post-processing capability exist (see Sec. 7.9) or are planned, and NASTRAN runs as well as plots are used in the batch mode. Cost estimates for plotting are missing from the documentation; plotting costs can be considerable.

## 5.9. Restart Capability

NASTRAN contains a sophisticated restart capability. Restarts are effective for: (1) continuing problem execution having an unscheduled interruption caused by a data error; (2) requesting additional information for a problem already completed; (3) running additional load cases in static analysis; and (4) extracting real eigenvalues for additional frequency ranges in normal modes analysis. Restarts are ineffective when the problem is small, or when changes are made in the element properties or grid point information. A CHKPNT YES card in the Executive Control Deck will save information on a new problem tape NPTP, subsequently renamed an old problem tape OPTP in a future run. One disadvantage of using restarts is the necessity of maintaining checkpoint files.

## 5.10. Nonlinear Capabilities

The nonlinear capabilities in NASTRAN are limited. A rough guess is that probably less than 5 percent of all NASTRAN users has ever attempted the nonlinear usage. Two types of static nonlinearities are offered: differential stiffness and material plasticity. The geometric stiffness effects treated by Rigid Formats 4 and 5 provide the user with a second order approximation to the nonlinear effects of large deflection. Rigid Format 6 offers a piecewise linear analysis capability (material plasticity). Only isotropic materials are allowed, and the capability is restricted to certain elements: ROD, TUBE, BAR, and plate elements (based on 2-D plasticity theory by J. L. Swedlow). The material properties are assumed to be stress dependent. The stiffness matrix is assumed constant over each load increment. After each increment, a new stiffness matrix is generated based on the current state of stress in each element. All static load options are allowed, except temperature and enforced element deformations.

For transient analysis, the module TRD provides for four types of nonlinear elements, mainly for control systems simulation. Nonlinear effects are treated as an additional applied load vector, whose elements are functions of either displacements or velocities. One additional nonlinear capability is nonlinear steady-state heat transfer analysis, which allows for: temperature-dependent conductivities for the elements, nonlinear radiation exchange, and a limited use of multipoint constraints. Nonlinear elements such as gaps and friction elements do not exist in COSMIC/NASTRAN. The NASTRAN nonlinear capabilties offered are primitive when compared to those of current nonlinear codes such as ADINA, AGGIE, ABACUS, ANSYS, and MARC.

## 5.11. Special NASTRAN Features

NASTRAN contains certain special or unique features not commonly found in most general purpose codes. These are briefly surveyed in Table 5-5. In general, the most likely users of these features are the expert/experienced ones. The common and novice users will probably find most of these features too difficult to use. These features represent a considerable investment in time and money, and are no doubt valuable to certain small pockets of the user community. However, their overall effectiveness and extent of usage are difficult to assess, and were not evaluated in the study.

## 5.12. Substructuring

The automated multi-stage substructuring system (Ref. 7, 11, 27) in NASTRAN, developed by Universal Analytics, Inc., is very powerful yet flexible and user-oriented. It incorporates an automated multi-stage modal synthesis procedure (Ref. 28) for the dynamic analysis of very large models, and a multi-stage component mode synthesis method (Ref. 29). Table 5-6 compares the NASTRAN automated multi-stage substructuring capability versus that offered by ANSYS. Both codes have highly-developed substructuring capabilities which have been used widely. Similarities and differences are seen in the design philosophy of the two codes and both advantages and disadvantages are noted. Substructuring is suggested only for the expert/experienced users. A learning curve is invariably necessary. However, for certain classes of problems, substructuring offers tremendous benefits and savings.

TABLE 5-5. SPECIAL NASTRAN FEATURES

| FEATURE | DESCRIPTION |
|---|---|
| 1. Static analysis with inertia relief | .Treats structural models not fully constrained.<br>.Computes inertia effects of unconstrained rigid body accelerations |
| 2. Manual single-stage substructure analysis | .Phase I analyze each substructure, establish master degrees of freedom<br>.Phase II analyze "pseudo-structure"<br>.Phase III back substitute for substructure displacements and stresses<br>.Can be used for all rigid formats except piecewise linear analysis |
| 3. Fully stressed design optimization (Rigid Format 1 - Linear Statics)<br>(Ref. 11 Sec. 26) | .Very simple design algorithm to resize elements<br>.For each design iteration, change each element cross-sectional properties to get a limit stress somewhere within element<br>.Procedure works reasonably well for statically determinate structures, acceptable for slightly indeterminate, and poor for indeterminate structure with high redundancy |
| 4. Cyclic Symmetry<br>(Ref. 7,11,40) | .Linear problems only; static and normal modes analyses<br>.Uses finite Fourier transformation<br>.Symmetric with respect to axis: rotational, dihedral symmetry<br>.Model 1 segment; use uncoupled, transformed equations |
| 5. Component Mode Synthesis<br>(Ref. 29) | .Design tool - evaluate critical component modes in total response<br>.Divide structure into separate components, reduce order of component matrices, combine them using compatibility at common boundary points |
| 6. Automated multi-stage substructure analysis<br>(Ref. 11,27,28) | .3 phases like "2", substructure operating file (SOF) created<br>.User can repeatedly combine and reduce structures<br>.Linear static and dynamic analyses<br>.Sequence control now automated using Substructure Case Control deck |
| 7. Representation of part of a structure by its vibration modes | .Use modal information from other analyses or vibration test data<br>.3 cases: all connection coordinates free, restrained, or mixed<br>.Restrictions: linearity, conservation of energy, reciprocity |

TABLE 5-5 cont'd.

| FEATURE | DESCRIPTION |
|---|---|
| 8. Representation of control systems | .Linear and nonlinear control systems simulated<br>.Properties of control system treated in quadratic format like in dynamics: $Mp^2 + Bp + K$ |
| 9. Structure/fluid interaction (Hydroelastic analysis) | .Compressible fluids in axisymmetric tanks<br>.Compressible fluids in rotationally symmetrical cavities with slots |
| 10. Acoustic cavity analysis | .For each element, pressure field assumed to vary linearly over the cross section and sinusoidally around the axis in circumferential direction.<br>.Slot portion of cavities limited to certain shapes |
| 11. Aeroelastic analysis | .Can be used in conjunction with structural analysis<br>.2 rigid formats: modal flutter analysis (by 3 methods); modal transient and frequency response analysis of aeroelastic models |
| 12. Special elements | .General element GENEL .Aeroelastic elements AEROi<br>.Dummy elements CDUMi .Acoustic elements AXIFi<br>.Rigid elements RIGDi .Hydroelastic elements FLUIDi |

TABLE 5-6. COMPARISON OF NASTRAN AUTOMATED MULTI-STAGE SUBSTRUCTURING vs ANSYS SUPERELEMENT CAPABILITY

| | NASTRAN | ANSYS |
|---|---|---|
| A. Major phases | Phase 1. Analyze each substructure to produce a description of its properties at boundary degrees-of-freedom "$u_a$". Restrictions:<br>.Statics, normal modes only<br>.No piecewise linear analysis<br>.All points on boundaries to be joined must be included<br>.Internal gridpoint identification must be in same order<br>Initial generation of individual basic matrices. Several runs performed, one for each substructure. | Substructure Generation Pass:<br>Creates reduced stiffness mass, and/or damping matrices, and load vectors of superelements, master degrees of freedom (MDOF's) selected. Restrictions:<br>.Linear elements<br>.Constant material properties<br>.No convection B.C.'s in load vector |
| | Phase 2. Analyze "pseudostructure",assembly and solution, recovery:<br>.1 or more executions<br>.Any number of substructure reductions and/or combinations<br>.Can be stopped at any stage, restarted<br>.Obtain $u_a$ vector | Superelement Use Pass:<br>Superelement STIF 50 used like any other ANSYS element; solved at MDOF's; degree-of-freedom directions are fixed to the element |
| | Phase 3. Analyze each substructure using $u_a$, complete data recovery for displacements and stresses in each individual basic substructure | Superelement Stress Pass:<br>Use MDOF's into each superelement to recover a full set of displacements (or temperatures) and stresses (or heat flows) |
| B. Reduction Technique | Static: Guyan reduction<br>Modal synthesis: Modal reduction | "Dynamic matrix condensation" technique (or Guyan reduction) |

TABLE 5-6 cont'd.

| | NASTRAN | ANSYS |
|---|---|---|
| C. User Control | Automated: substructure control deck required | User keeps track of tape numbers in each pass |
| D. Advantages | 1. Automated data base management system: substructure operating file, which can be edited . | 1. Separates linear from nonlinear portions. |
| | 2. Data transfer possible among IBM, UNIVAC, and CDC computers at any stage of analysis . | 2. Separates "fixed design" portion from portion undergoing design change . |
| | 3. Model only 1 of two or more identical substructures , superelement tree. | 3. Transferable matrices for different analysis types within ANSYS . |
| | 4. Dry run for data deck; STEP option . | 4. Capability for "reflected superelements", symmetry, hierarchy. |
| | 5. Symmetry, repeatability, and hierarchy possible. | 5. Automated selection of master degrees of freedom (Rev. 3 Update 67, 1979). |
| | 6. Automatically generated DMAP alters . | |
| | 7. No restrictions on grid point and element numbering. | |
| E. Disadvantages | 1. No heat transfer, nonlinear, or transient response. | 1. New learning curve and terminology. |
| | 2. Difficult for novice; learning curve. | 2. MDOF's can limit problem size. |
| | 3. Reduced computer-to-computer mobility. | 3. More file handling and more runs required. |

# SECTION 6

## VERIFICATION EXERCISES

In the course of reviewing the copious NASTRAN documentation, the authors were very impressed with the complexity of the problems solved using NASTRAN. However, it is often difficult to determine the accuracy of such complex solutions as there are often no theoretical solutions available. Experimental results are limited and it is difficult to obtain the idealized loading and boundary conditions assumed in analysis. Therefore, a limited number of structural verification problems were run. The problems selected could be easily verified against "textbook" solutions.* These examples are included in this section.

A number of interesting observations can be made based upon these results. Example 1 indicates that the accuracy of the code is in good agreement with the closed form solution for this problem. Example 2 illustrates the effect that MPC's can have on the solution for certain elements. Example 3 was chosen to exercise the various options available in NASTRAN for eigenvalue extraction. All were in good agreement when a consistent mass matrix was used.

---

*The solutions to these problems using the ANSYS general purpose computer program are available from the open literature and are included for comparison.

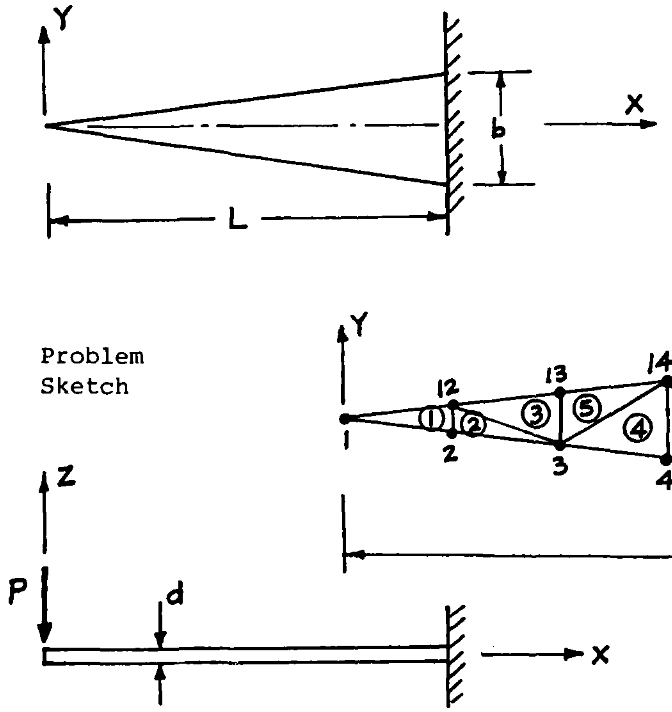
# EXAMPLE 1. BENDING OF A TAPERED PLATE

Type: Static analysis, plate elements

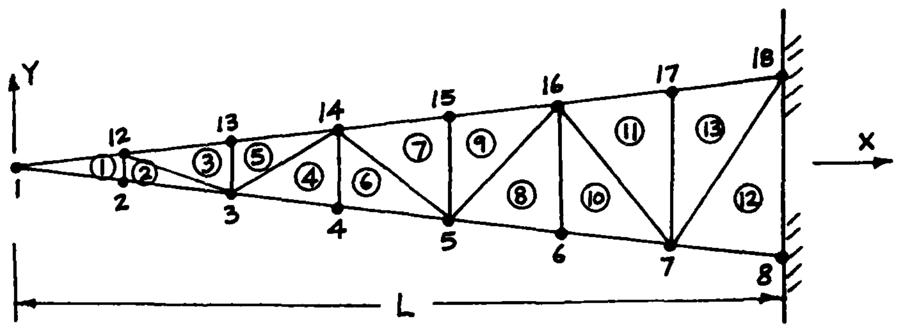
Reference: Harris, C.O., Introduction to Stress Analysis, The MacMillan Co., New York, 1959, Page 114, Problem 61.

Problem: A tapered cantilever plate of rectangular cross-section is subjected to a load P at its tip. Find the maximum deflection $\delta$ and the stress $\sigma_x$ in the plate.

Given: $E = 30 \times 10^6$ psi, $L = 20$ in., $b = 3$ in., $d = 0.5$ in., $P = 10$ lb.




Problem Sketch




Finite Element Model

Results:

| | $\delta$ (in.) | $\sigma_x$ (psi) |
|---|---|---|
| Theory | -0.0426666 | 1600. |
| ANSYS (STIF 6) (Ref. 30, Problem 34) | -0.0426668 | 1600. |
| NASTRAN ( TRIA2) | -0.0426672 | 1625. |

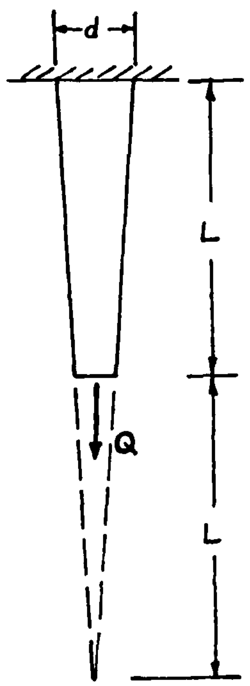
## EXAMPLE 2. ELONGATION OF A SOLID BAR

**Type:** Static analysis, solid elements

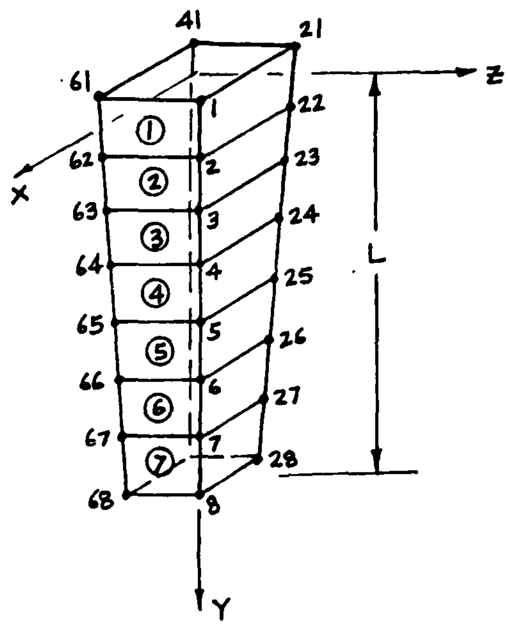
**Reference:** Harris, C.O., Introduction to Stress Analysis, The MacMillan Co., New York, 1959, Page 237, Problem 4.

**Problem:** A tapered bar of square cross-section, made of an aluminum alloy for which E is 10, 400,000 psi, is suspended from a ceiling. An axial load Q is applied to the end of the bar. Determine the maximum axial deflection $\delta$ in the bar and the axial stress $\sigma_y$ at mid-length.

**Given:** L = 10 in., d = 2 in., Q = 10,000 lb.




Problem Sketch




Finite Element Model

**Results:**

| | $\delta$ (in.) | $\sigma_y$ (psi) | No. of input cards |
|---|---|---|---|
| Theory | 0.00480769 | 4444. | - |
| ANSYS (Ref. 30, Problem 37) (STIF 5) Difference | 0.00478878<br>0.39% | 4441.<br>0.067% | 33 |

EXAMPLE 2

Results cont'd.

| | | $\delta$(in.) | $\sigma_y$(psi) | No. of input cards |
|---|---|---|---|---|
| NASTRAN: | HEXA1 with MPC's | 0.00479047 | 4441. | 91 |
| | Difference | 0.36% | 0.067% | |
| Without MPC's to enforce symmetry: | | | | |
| 1. HEXA1 | (Nodes 28,68) | 0.006179 | 4441. | 67 |
| | (Nodes 6,48) | 0.007493 | | |
| | Difference | 30.0%/56.4% | .067% | |
| 2. IHEX1 | | 0.00479047 | 4441. | 67 |

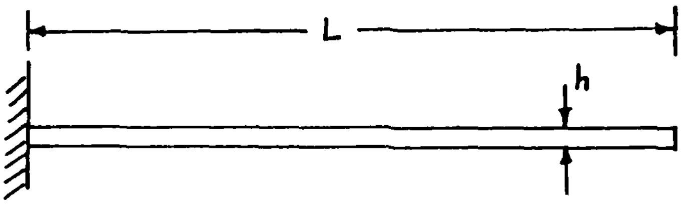
## EXAMPLE 3. NATURAL FREQUENCIES OF A CANTILEVER BEAM

Type: Normal mode analysis (eigenvalue extraction), beam elements

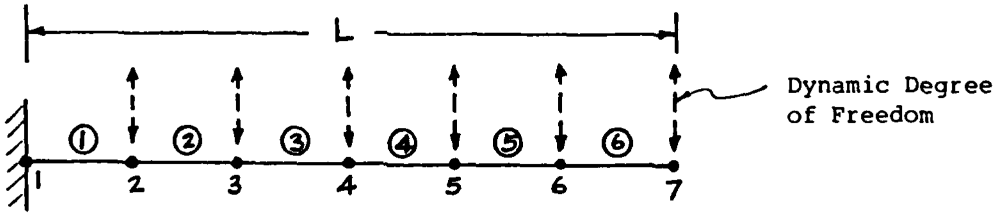
Reference: Thomson, W. T. Vibration Theory and Applications, Prentice-Hall, Inc. Englewood Cliffs, N. J., 1965, Pages 275 and 357.

Problem: Determine the first three natural frequencies $f_i$ of a uniform beam clamped at one end and free at the other end.

Given: $E = 30 \times 10^6$ psi, $I = 1.3333$ in$^4$, $A = 4$ in$^2$, $h = 2$ in., $L = 80$ in., $w = 1.124$ lb/in.




PROBLEM SKETCH




Finite Element Model
(Uses 6 lateral dynamic degrees of freedom)

Results:

| | $f_1$(Hz) | $f_2$(Hz) | $f_3$(Hz) |
|---|---|---|---|
| Theory | 10.247 | 64.221 | 179.82 |
| ANSYS (Ref. 30, Problem 51) | 10.247 | 64.197 | 180.14 |
| (STIF 3) Difference | None | 0.037% | 0.178% |
| NASTRAN (BAR) | | | |
| 1. Lumped mass model | 10.105 | 61.437 | 167.542 |
| 2. Inverse power method-consistent mass | 10.203 | 64.145 | 179.905 |
| 3. Determinant method | 10.233 | 64.148 | 179.905 |
| 4. Givens method | 10.233 | 64.148 | 179.905 |
| 5. Unsymmetrical inverse power | 10.233 | 64.148 | 179.905 |
| 6. Unsymmetrical determinant | 10.233 | 64.148 | 179.905 |

# SECTION 7

# ADVANCED EVALUATION

The advanced evaluation consisted of a comprehensive literature survey on a selected group of NASTRAN finite elements and performing a series of computer runs aimed at assessing element quality, convergence, completeness, and efficiency. As much as possible, the framework of this evaluation is based on comparisons with other well-known benchmark tests, and on recent publications on simple element evaluation tests and criteria (References 31 to 35). The intent of this section is to document in one place what is currently known about the quality of some representative COSMIC/ NASTRAN structural elements:

QDPLT, TRSHL, TRIM6, TRPLT1, CONEAX, TRAPRG, TRAPAX, HEXA1, HEXA2, IHEX1 and IHEX2.

After these element evaluation results are described, we will also briefly discuss current minicomputer versions of NASTRAN, commercially available pre- and post-processors for NASTRAN, and a study that was conducted on trends of run cost and CPU time versus mesh sizes.

## 7.1. Advanced Evaluation Tests

To aid in the overall assessment of any finite element, Robinson has suggested several evaluation tests and the following assessment points (Ref. 31) before "the element is turned loose in the commercial arena":

1. Element shape.
2. Basic assumption (stress, displacement, or mixed).
3. Number of nodes.
4. Degrees of freedom, types of freedom, and number of freedoms per node.
5. Are the freedoms conventional and understandable?
6. Does the element pass the standard and recognized tests?
7. How general is the element in its application?
8. Are the assumed stress or displacement functions simple?
9. Is the derivation of the element matrices simple, straightforward and understandable, and are the matrices given in explicit form?
10. What element matrices are available?
11. Is the element a synthesized element, that is, is it built-up of a number of elements?

12. Are the element matrices easy to code?
13. Is the element theory and coding well-documented together with check examples?
14. Does the element contain any false zero energy modes (false zero stress or strain states) other than those corresponding to rigid body modes?
15. Is the element mixable with other elements within a particular program?
16. Can the element be easily incorporated into a particular program?
17. Has the element been fully checked out? How often and in what problems has it been used? The debugging of an element is very difficult and is made even more difficult by element complexity.
18. Is the originator of the element available for consultation if problems arise? In many cases, the software developers are unaware of the detailed theory behind their elements.
19. What are the element material properties?
20. Does the element help to reduce the effort to prepare the model, to obtain computer results, to interpret the results?
21. For a given model, do the results change if the element specifying nodes are written in a different order?

Table 7-1 is our assessment of how COSMIC/NASTRAN rates in these categories for the selected group of elements. This assessment is, in many cases, quite subjective and is based largely on the documentation (primarily the Theoretical Manual). Results show that in general, according to Robinson's criteria, COSMIC/NASTRAN rates a "fair" ranking in its documentation for any particular element.

Robinson also recommended these basic evaluation tests (References 31,34):

- Single element test
- Convergence test
- Element completeness test
- Patch test
- False zero energy mode test

The "single element test" consists of taking a single element in rectangular form, considering it as a structure and then investigating its behavior under various loading and constraint conditions for various geometries. This test is very helpful in understanding the load-carrying capabilities of the element and shows its sensitivity to aspect ratio. (This test is relatively simple to apply only for triangular or quadrilateral membrane and plate bending elements).

The "convergence test" consists of studying the behavior of some parameter or parameters for various numbers of elements in a model for a particular problem. As the number of elements in a model is increased, and the mesh becomes more refined, the results should approach the true solution.

The "element completeness tests" consist of showing that the element displacement/stress assumptions contain the constant strain/stress modes and the zero strain/stress (rigid body) modes. These are essential requirements for any finite element. If each element in a model passes these tests, and the model is a compatible/equilibrium one, the results will converge monotonically.

The "patch test" consists of applying a constant strain/stress state to an assemblage of elements and showing that all the elements contain the same constant strains/stresses. This test is for "nonconforming" or "noncompatible" and "non-equilibrium" models and should be passed if the results are to converge (though not necessarily monotonically). The patch test is a completeness condition for an assemblage of elements as against an individual element. The primary purpose of this test is to provide an estimate of the error which will be incurred. It is noted that increasing the number of elements in the "patch" will probably reduce the error if the element performs well in the convergence test.

ADVANCED EVALUATION OF NASTRAN STRUCTURAL ELEMENTS - POINTS OF ASSESSMENT

| QUESTION | QDPLT | TRSHL | CONEAX | TRAPRG | HEXA 1<br>HEXA 2 | IHEX1 | IHEX2 |
|---|---|---|---|---|---|---|---|
| 1. Element shape | Quadrilateral | Triangular | Conical Shell | Trapezoidal Ring | Brick, constant strain, 5 or 10 tetrahedra | Brick | Brick |
| 2. Basic assumption | DISPLACEMENT FORMULATION | | | | (constant strain) | (isoparametric) | (isoparametric) |
| 3. No. of nodes | 4 | 6 | 2 | 4 | 8 | 8 | 20 |
| 4. Degrees of freedom at each node | 1 transverse displacement<br>2 rotations | 3 translations<br>2 rotations | 3 translations<br>2 rotations | 3 translations | 3 translations | 3 translations | 3 translations |
| 5. Freedoms understandable and conventional? | Yes | Yes (No transverse shear flexibility) | Unique, incl. transverse shear flexibility | Yes | Yes | Yes | Yes |
| 6. Does element pass standard and recognized tests? | See Subsection 7.2 | Subsection 7.3 | Subsection 7.5 | Subsection 7.6 | Subsection 7.7 | Subsection 7.7 | Subsection 7.7 |
| 7. How general is element in its application? | General | General, thin, shallow shell element | Axisymmetric but loads and deflections do not have to be | Must have 2 parallel sides, restrictive use. No input option for pressure on an element face. | General, but requires too many elements for accuracy | General; good for problems with high shear stresses | General; good for plate bending type deformations |

TABLE 7-1

TABLE 7-1 (cont'd)

| | QDPLT | TRSHL | CONEAX | TRAPRG | HEXA1 HEXA2 | IHEX1 | IHEX2 |
|---|---|---|---|---|---|---|---|
| 8. Assumed displacement functions simple? | Yes, $x^2y$ term omitted | No; quadratic polynomial for membrane, quintic for w | Yes | Yes, harmonic expansion in circumferential direction | Yes | Linear displacement variation | Quadratic displacement variation |
| 9. Element matrices-simple, understandable, given explicitly? | Yes | Not given | No, could be better | Not given explicitly | No | No | No |
| 10. Element matrices available? | Yes | No | Yes, to an extent | No | No | No | No |
| 11. Built-up element? | Yes, 4 basic TRPLTs | Yes, constituents are TRIM6 and TRPLT1 | No | No | Yes, 5 or 10 overlapping tetrahedra | No | No |
| 12. Element matrices easy to code? | Yes | No | No | No | Yes | No | No |
| 13. Element theory well-documented? Check examples? | Yes | Fair, no examples | Yes, but no examples | Fair; is modified Wilson element | No; poor documentation, no examples | Poor, no examples | Poor, no examples |
| 14. Any false zero energy modes? | UNDOCUMENTED | | | | | | |
| 15. Element mixable with others in code? | Yes | Yes | No | No | Yes | Yes, except with special stand-alone elements | Yes, except with special stand-alone elements |

TABLE 7-1 (cont'd.)

| QUESTION | QDPLT | TRSHL | CONEAX | TRAPRG | HEXA1 HEXA2 | IHEX1 | IHEX 2 |
|---|---|---|---|---|---|---|---|
| 16. Element easily incorporated into code? | Yes | Yes | Yes | Yes | Yes | Yes, coding of numerical integration important | Yes, coding of numerical integration important |
| 17. Element fully debugged? | Yes, except for a twist load | No | Yes, but thermal load capability reported to be in error | Yes | Yes, but poor in bending problems | Yes | Yes |
| 18. Element originator available for consultation? | No | Yes, but Narayanaswami now at MSC | No | No | No | No | No |
| 19. Element material properties | Isotropic, anisotropic | Isotropic only | Isotropic only | Isotropic, orthotropic | Isotropic only | Isotropic only | Isotropic only |
| 20. Element helps to reduce model preparation effort, obtain & interpret results? | Yes | Yes, higher order, but results questionable (see Fig. 7.3-2) | Yes, good performance even with a few elements | Yes, except for trapezoidal restriction and no pressure input option | No, too many elements required for satisfactory accuracy | Yes, 8-node element easy to use | No, 20-node element requires more cumbersome input |
| 21. Results change if node ordering different? | Not evaluated in this study | Not evaluated in this study | No | Not evaluated in this study | HEXA1: Yes<br>HEXA2: No | Not evaluated in this study | Not evaluated in this study |

The "false zero energy mode test" consists of showing that an element does not contain zero strain/stress fields other than those associated with rigid body modes. Elements can contain false zero states which are due to the basic strain/stress assumptions and the subsequent choice of nodal degrees of freedom.

In addition to these tests, we will often include the results on "element efficiency test". This test will in general be different for each type of element, and shows a measure of element performance and coding efficiency for that problem. Some well-known benchmark cases used by many researchers include:

- .Plate bending element. Simply supported, or clamped, square plate loaded centrally by a concentrated load.
- .Plate/shell element. Scordelis-Lo cylindrical shell problem loaded by its own weight.
- .Conical shell element. Cantilevered cylindrical shell loaded at its free end by a line load around the circumference.
- .Solid of revolution element. Thick-walled cylinder loaded by internal pressure.
- .3-D solid elements. Slender cantilever beam problem, a pressurized thick-walled cylinder problem, and a study of element eigenvalues and eigenvectors.

At the end of each element evaluation, an assessment will be made of that element's quality and efficiency.

## 7.2. The QDPLT Element

QDPLT was selected for advanced evaluation because it is one of the more complex plate bending elements in the NASTRAN library. Also, there is less documentation in the Theoretical Manual for this element than for most other shell elements. In the course of this investigation, it was determined that deficiencies had been previously observed in QDPLT (Ref. 33). QDPLT consists of four nonconforming TRPLT's, models bending behavior only, and does not have membrane-bending coupling.

### 7.2.1. Single Element Test

The single element test for a plate bending element consisted of evaluating the displacement and rotation for eight cases (giving various combinations of constraints and loads) for a range of element aspect ratios:

Cases 1, 2, 3, and 4: Displacement in the z-direction ($U_z$) and rotation about the y-axis at node 3 ($\Theta_y$).

Cases 5,6,7, and 8: Displacement in the z-direction ($U_z$) and rotation about the x-axis at node 3 ($\Theta_x$).

The single element test results are plotted in Figures 7.2-1 to 7.2-8, where they are compared with the standard beam solutions.

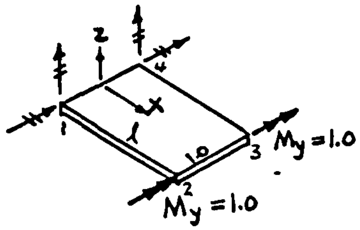
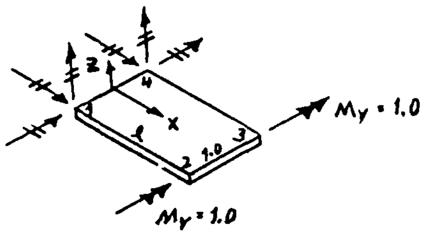
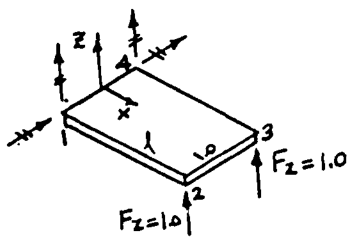
For our evaluation of QDPLT, the following material properties were used:

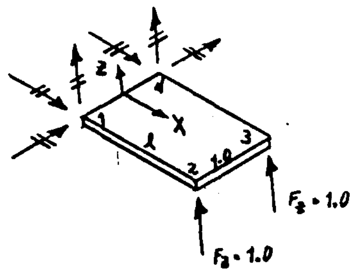
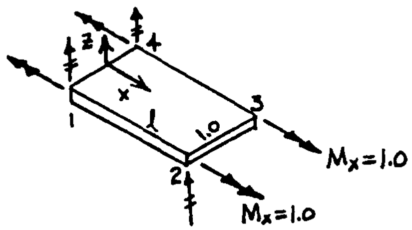
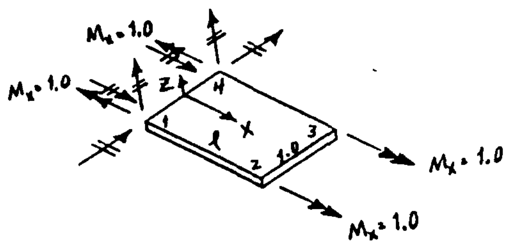
Young's modulus $E = 10^7$ psi

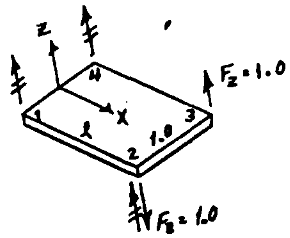
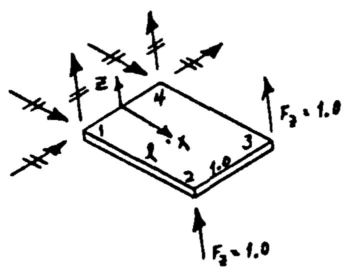
Shear modulus $G = \dfrac{E}{2(1+\nu)} = 3.8462 \times 10^6$ psi

Poisson's ratio $\nu = 0.3$

For example, the theoretical values are presented on the following three pages:

| CASE | | BOUNDARY CONDITIONS AT SUPPORTED EDGE | THEORETICAL RESULTS AT FREE END |
|---|---|---|---|
| 1 |  | $(\Theta_y)_1 = (\Theta_y)_4 = 0$<br>$(U_z)_1 = (U_z)_4 = 0$ | $\Theta_y = \frac{M_y \ell}{EI} = 2.40 \times 10^{-3} \cdot \ell$<br>$U = \frac{M_y \ell^2}{2EI} = 1.20 \times 10^{-3} \cdot \ell^2$ |
| 2 |  | $(\Theta_y)_1 = (\Theta_y)_4 = 0$<br>$(\Theta_x)_1 = (\Theta_x)_4 = 0$<br>$(U_z)_1 = (U_z)_4 = 0$ | $\Theta_y = \frac{M_y \ell (1-\nu^2)}{EI} = 2.18 \times 10^{-3} \cdot \ell$<br>$U_z = \frac{M_y \ell^2 (1-\nu^2)}{2EI} = 1.09 \times 10^{-3} \cdot \ell^2$ |
| 3 |  | $(\Theta_y)_1 = (\Theta_y)_4 = 0$<br>$(U_z)_1 = (U_z)_4 = 0$ | $\Theta_y = \frac{F_z \ell^2}{2EI} = 1.20 \times 10^{-3} \cdot \ell^2$<br>$U_z = \frac{F_z \ell^3}{3EI} = .800 \times 10^{-3} \cdot \ell^3$ |

| CASE | | BOUNDARY CONDITIONS AT SUPPORTED EDGE | THEORETICAL RESULTS AT FREE END |
|---|---|---|---|
| 4 |   | $(\Theta_y)_1 = (\Theta_y)_4 = 0$<br>$(\Theta_x)_1 = (\Theta_x)_4 = 0$<br>$(U_x)_1 = (U_x)_4 = 0$ | $\Theta_y = \dfrac{F_z \ell^2 (1-\nu^2)}{2EI} = 1.09 \times 10^{-3} \cdot \ell^2$<br>$U_z = \dfrac{F_z \ell^3 (1-\nu^2)}{3EI} = .780 \times 10^{-3} \cdot \ell^3$ |
| 5 |   | $(U_z)_1 = (U_z)_2 = (U_z)_4 = 0$ | $\Theta_y = \dfrac{T\Delta Y}{GJ} = .780 \times 10^{-3}$<br>$\Theta_x = \dfrac{M_x \ell}{GJ} = .780 \times 10^{-3} \cdot \ell$<br>$U_z = \dfrac{M_x \ell \Delta Y}{GJ} = .780 \times 10^{-3} \cdot \ell$ |
| 6 |   | $(\Theta_y)_1 = (\Theta_y)_4 = 0$<br>$(\Theta_x)_1 = (\Theta_x)_4 = 0$<br>$(U_z)_1 = (U_z)_4 = 0$ | $\Theta_y = \dfrac{T\Delta Y(1-\nu^2)}{GJ} = .710 \times 10^{-3}$<br>$\Theta_x = \dfrac{M_x \ell (1-\nu^2)}{GJ} = 1.42 \times 10^{-3} \cdot \ell$<br>$U_z = \dfrac{M_x \ell \Delta Y (1-\nu^2)}{2GJ} = .710 \times 10^{-3} \cdot \ell$ |

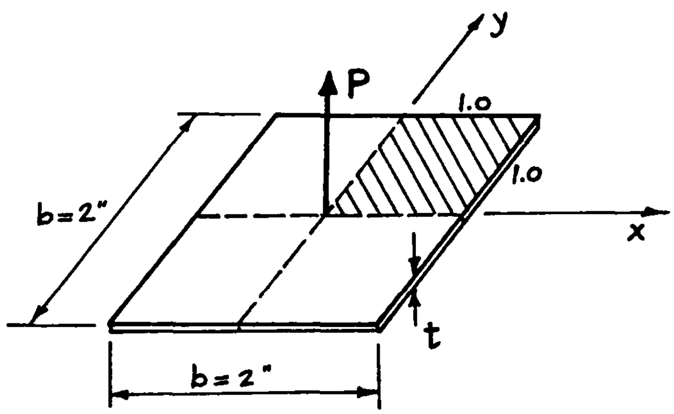
<table>
<tr><th>CASE</th><th></th><th>BOUNDARY CONDITIONS AT SUPPORTED EDGE</th><th>THEORETICAL RESULTS AT FREE END</th></tr>
<tr><td>7</td><td><br></td><td>$(U_z)_1 = (U_z)_2 = (U_z)_4 = 0$</td><td>$\Theta_y = \frac{T\Delta y}{GJ} = .780 \times 10^{-3}$<br>$\Theta_x = \frac{M_x \ell}{GJ} = .780 \times 10^{-3} \cdot \ell$<br>$U_z = \frac{M_x \ell \Delta x}{GJ} = .780 \times 10^{-3} \cdot \ell$</td></tr>
<tr><td>8</td><td><br></td><td>$(\Theta_y)_1 = (\Theta_y)_4 = 0$<br>$(\Theta_x)_1 = (\Theta_x)_4 = 0$<br>$(U_z)_1 = (U_z)_4 = 0$</td><td>$\Theta_y = \frac{T\Delta y(1-\nu^2)}{2GJ} = .355 \times 10^{-3}$<br>$\Theta_x = \frac{M_x \ell(1-\nu^2)}{GJ} = .710 \times 10^{-3} \cdot \ell$<br>$U_z = \frac{M_x \ell \Delta y(1-\nu^2)}{2GJ} = .355 \times 10^{-3} \cdot \ell$</td></tr>
</table>

The critical single element tests for QDPLT are Cases 6 and 8 (Figures 7.2-6 and -8). In both cases where the performance was poor, a twisting moment was applied to the element with one edge fully clamped. This activates differential (bilinear) bending. It is interesting to note that when the rotational restraint was not applied (Cases 5 and 7) with the loading otherwise identical, the element performed well.

Other 4-node plate bending elements, such as Lockheed/NASTRAN's LORA and MSC/NASTRAN's QUAD4 correlated reasonably well with theory up to aspect ratios of 12 (Ref. 33). Except for load cases 6 and 8, QDPLT performed well in the single element tests.

7.2.2. Convergence Test

A simple convergence test often used for plate bending elements is a square plate centrally loaded by a concentrated load P, with simply-supported and clamped boundary conditions:




Coefficients:

$$\alpha = \begin{cases} 0.1267 & \text{Simply supported} \\ 0.0611 & \text{clamped} \end{cases}$$

Theoretical deflections at the center are (Ref. 36):

$$\delta = \frac{\alpha P b^2}{E t^3}$$

Assume: $b = 2$ in. $\nu = 0.3$
$P = 4$ lb.
$t = 0.1$ in.
$E = 10^7$ psi

$$\delta = \begin{cases} 2.0272 \times 10^{-4} \text{ in.} & \text{simply-supported} \\ 9.976 \times 10^{-5} \text{ in.} & \text{clamped} \end{cases}$$

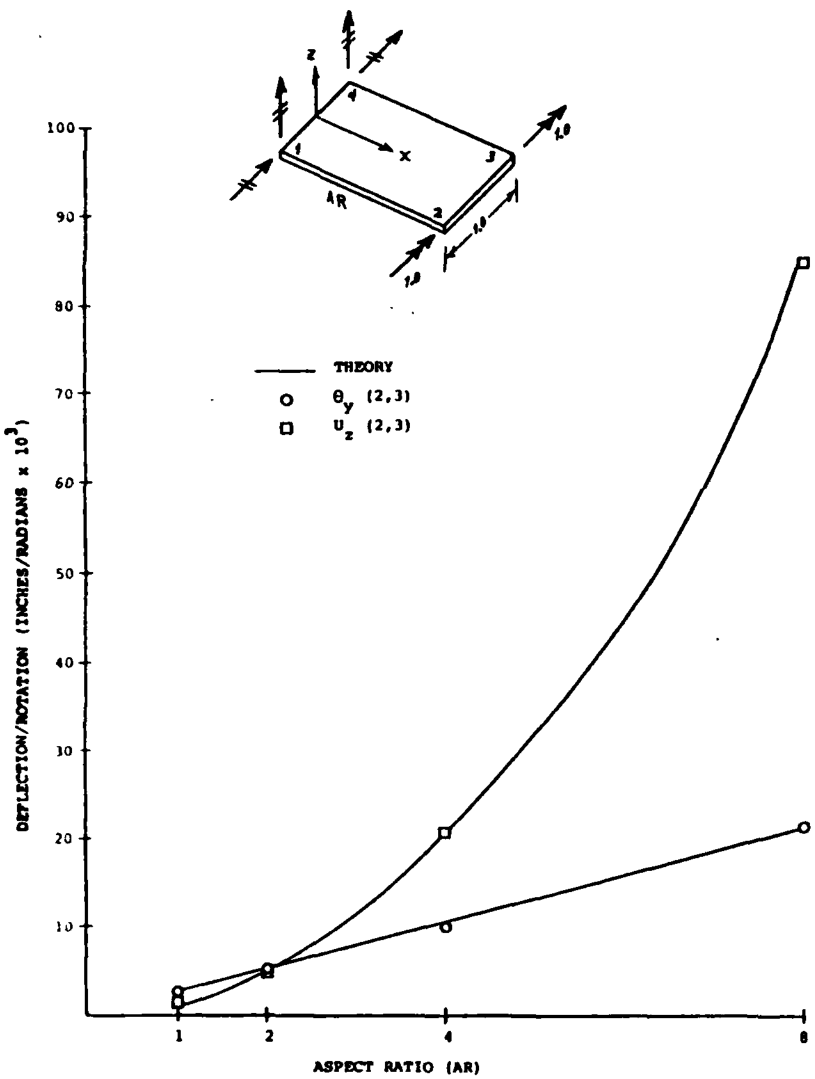



FIGURE 7.2-1. SINGLE ELEMENT TEST FOR QDPLT. CASE 1.

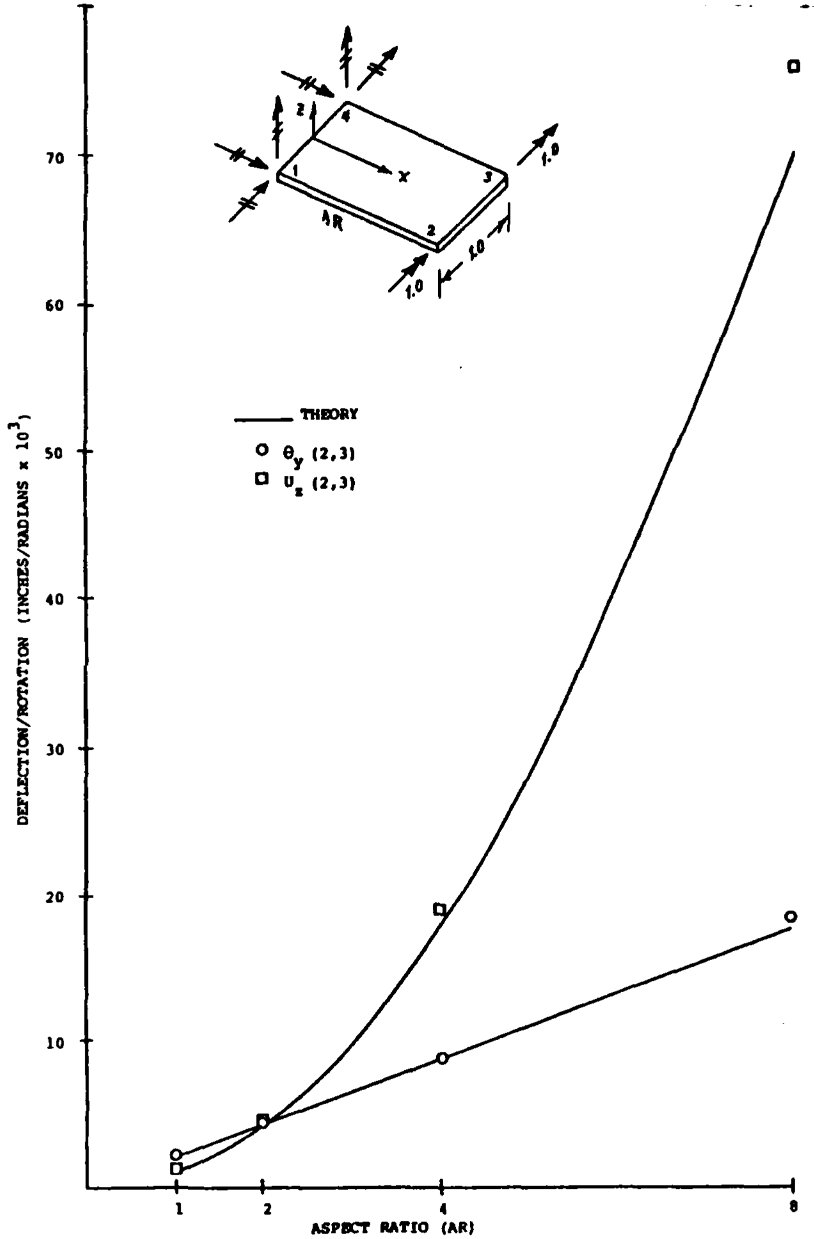



FIGURE 7.2-2. SINGLE ELEMENT TEST FOR QDPLT. CASE 2.

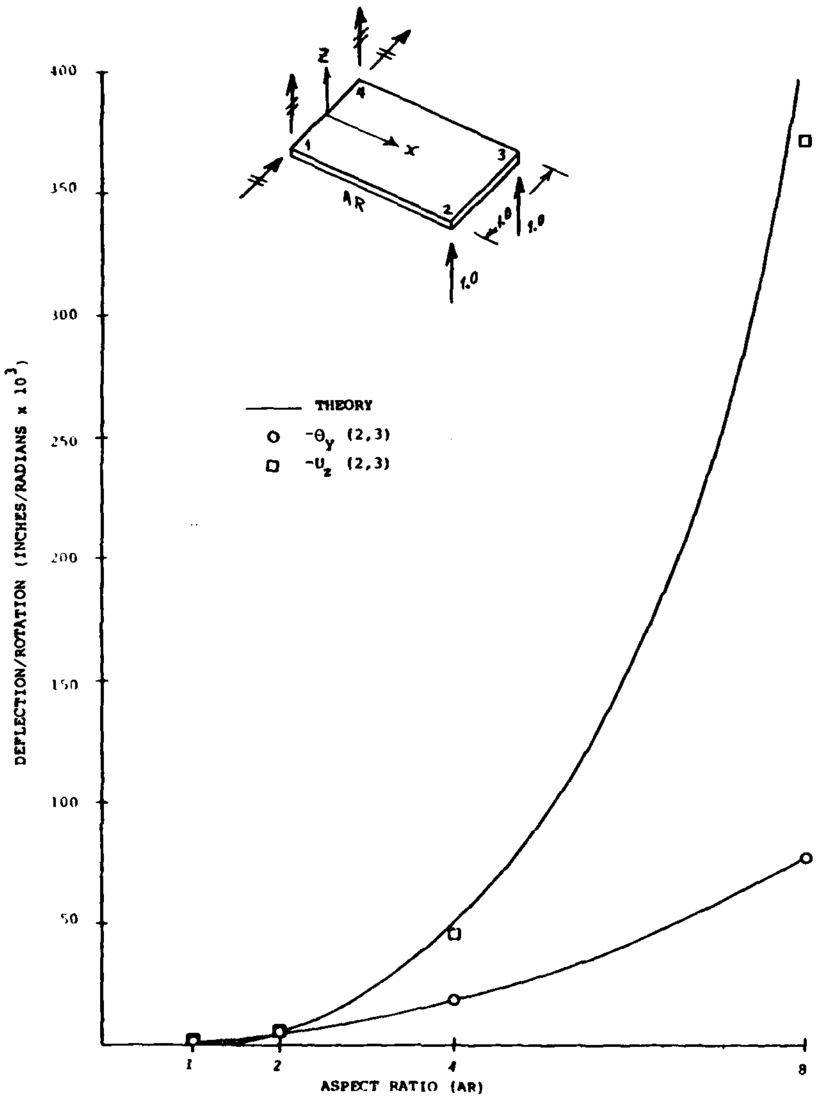



FIGURE 7.2-3. SINGLE ELEMENT TEST FOR QDPLT. CASE 3.

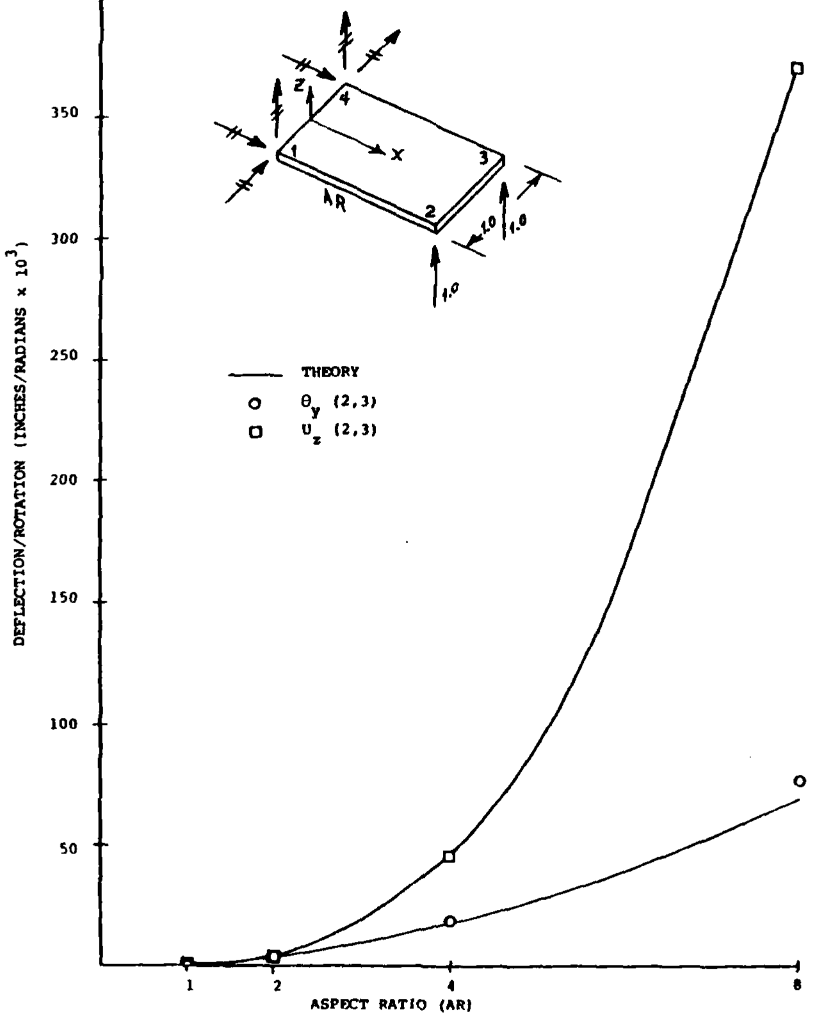



FIGURE 7.2-4. SINGLE ELEMENT TEST FOR QDPLT. CASE 4.

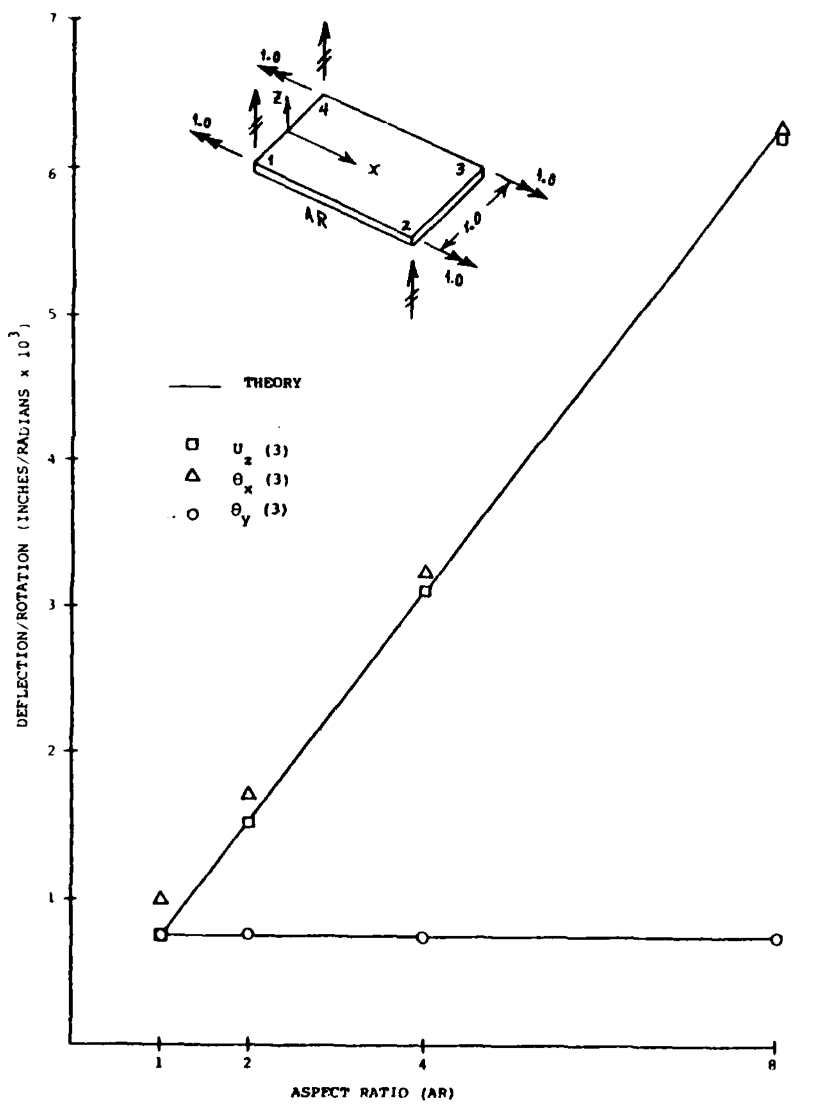



FIGURE 7.2-5. SINGLE ELEMENT TEST FOR QDPLT. CASE 5.

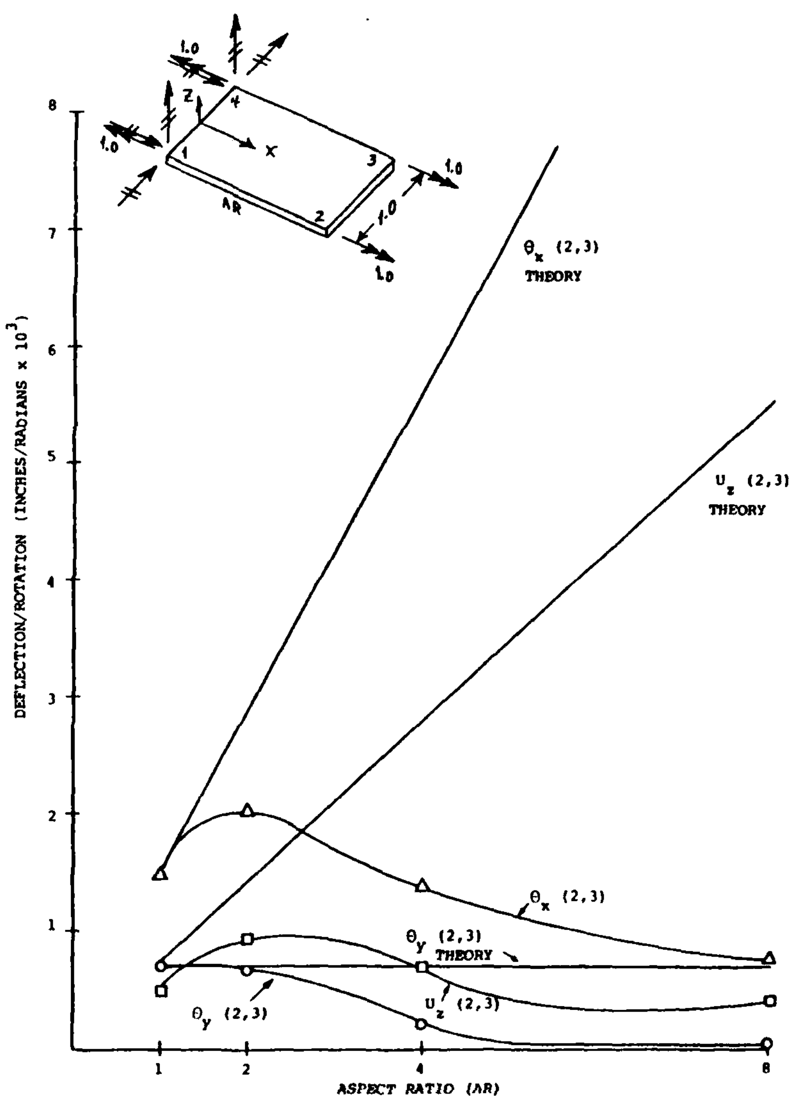



FIGURE 7.2-6. SINGLE ELEMENT TEST FOR QDPLT. CASE 6.

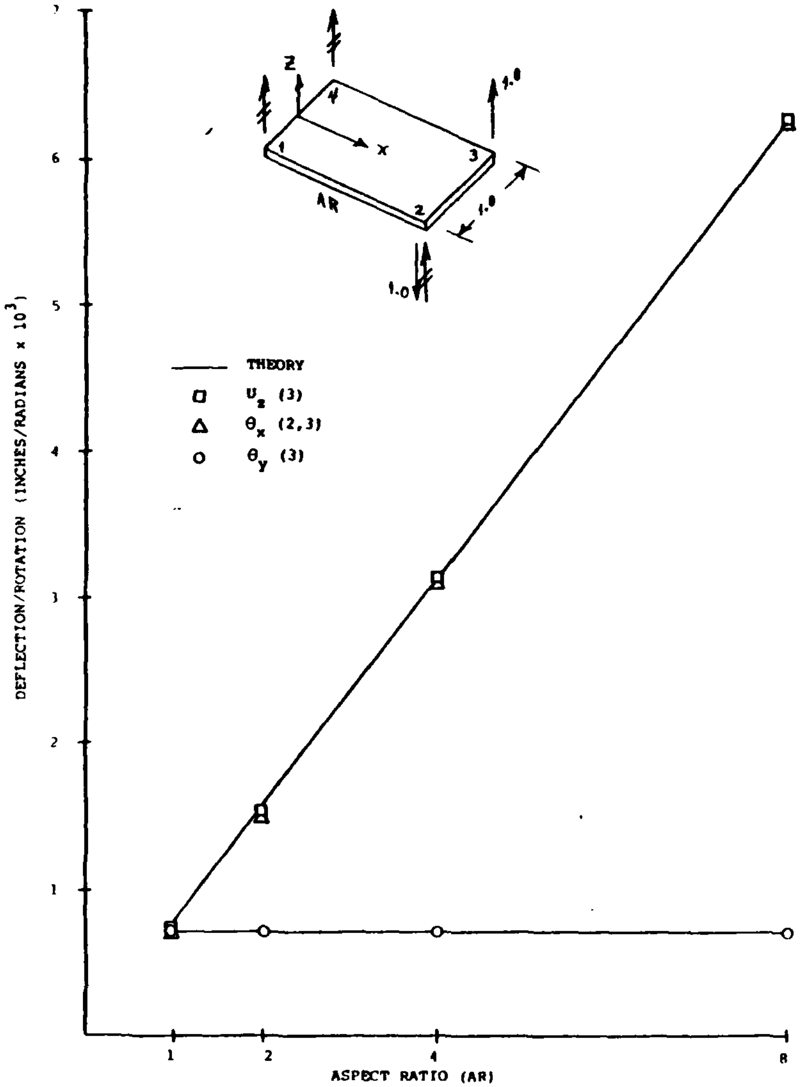



FIGURE 7.2-7. SINGLE ELEMENT TEST FOR QDPLT. CASE 7.

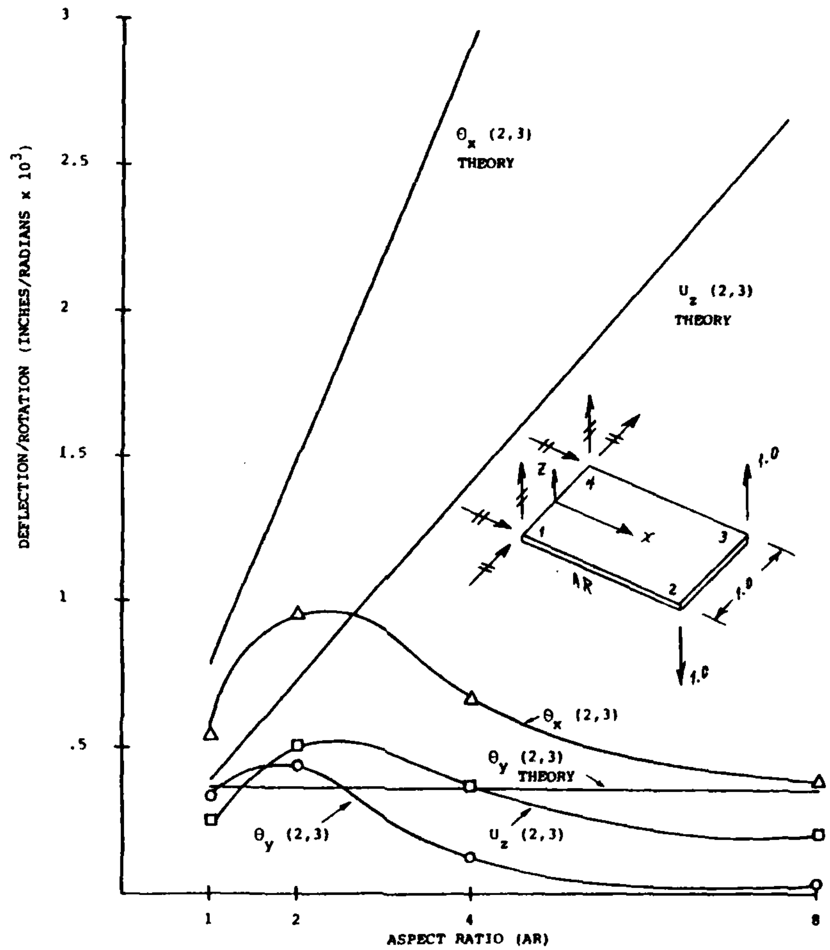



FIGURE 7.2-8. SINGLE ELEMENT TEST FOR QDPLT. CASE 8.

We evaluated QDPLT for four mesh sizes using this convergence test:

| NO. OF ELEMENTS FOR 1/4 OF PLATE | COMPUTED CENTRAL DEFLECTION (in.) | | | |
|---|---|---|---|---|
| | SIMPLY SUPPORTED | | CLAMPED | |
| | $\delta \times 10^4$ (in.) | % Error | $\delta \times 10^5$ (in.) | % Error |
| 1 | 2.198274 | 8.43 | 5.475623 | -42.3 |
| 4 | 2.134809 | 5.31 | 9.989985 | 0.140 |
| 9 | 2.109106 | 4.04 | 10.33296 | 3.57 |
| 16 | 2.100579 | 3.61 | 10.41435 | 4.39 |

These QDPLT results are plotted in Figures 7.2.2-1 and 7.2.2-2, and show the same convergence trends as in Ref. 31. Unfortunately, the modeling error discussion (Sec. 15.2) in Ref. 7 does not contain QDPLT results. It is interesting to note that the convergency trend is not monotonic and, in fact, appears to diverge slightly for the clamped case as the number of elements is increased, with four elements yielding very accurate results.

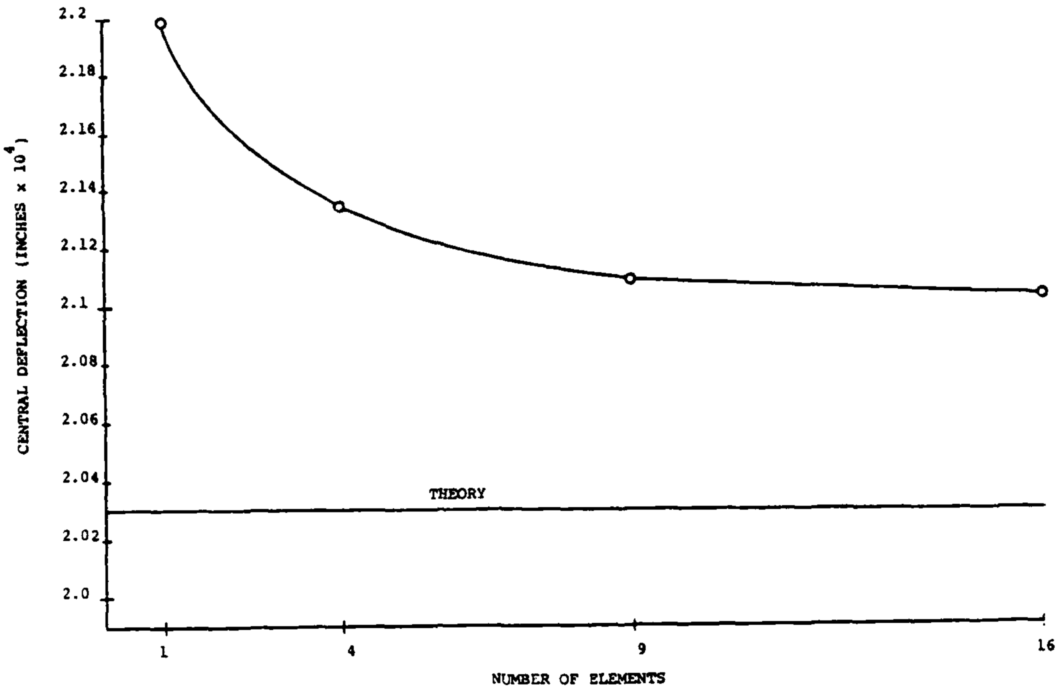



FIGURE 7.2.2-1. CONVERGENCE TEST FOR QDPLT. SIMPLY SUPPORTED SQUARE PLATE (AR = 1) WITH CENTRAL POINT LOAD.

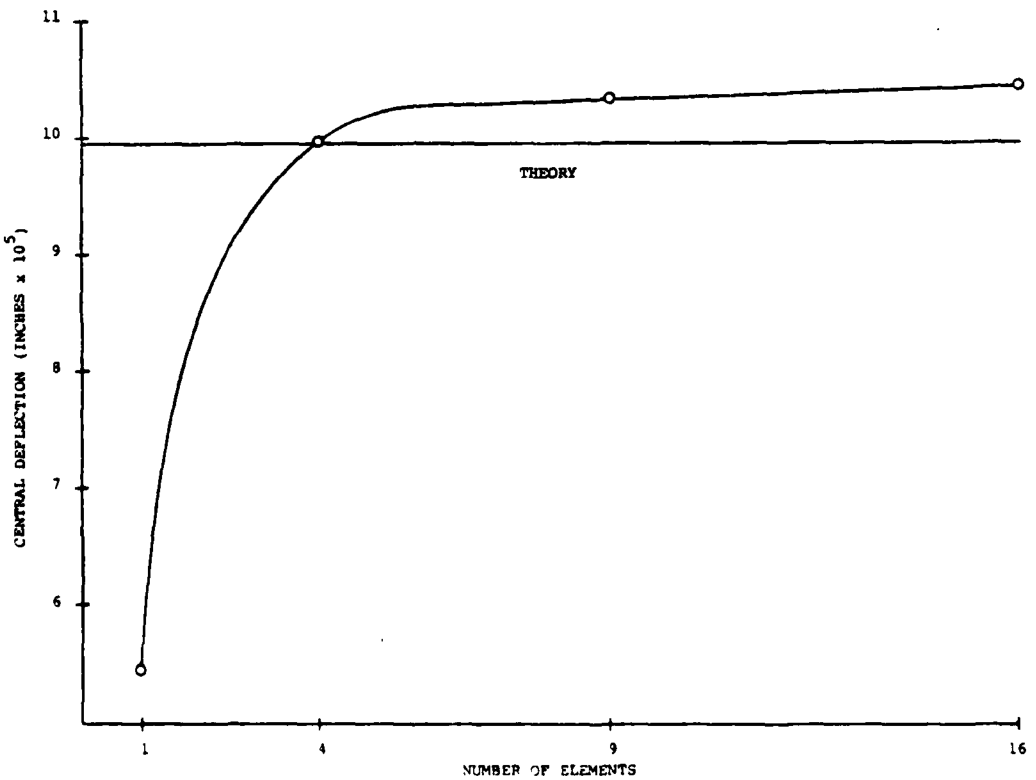
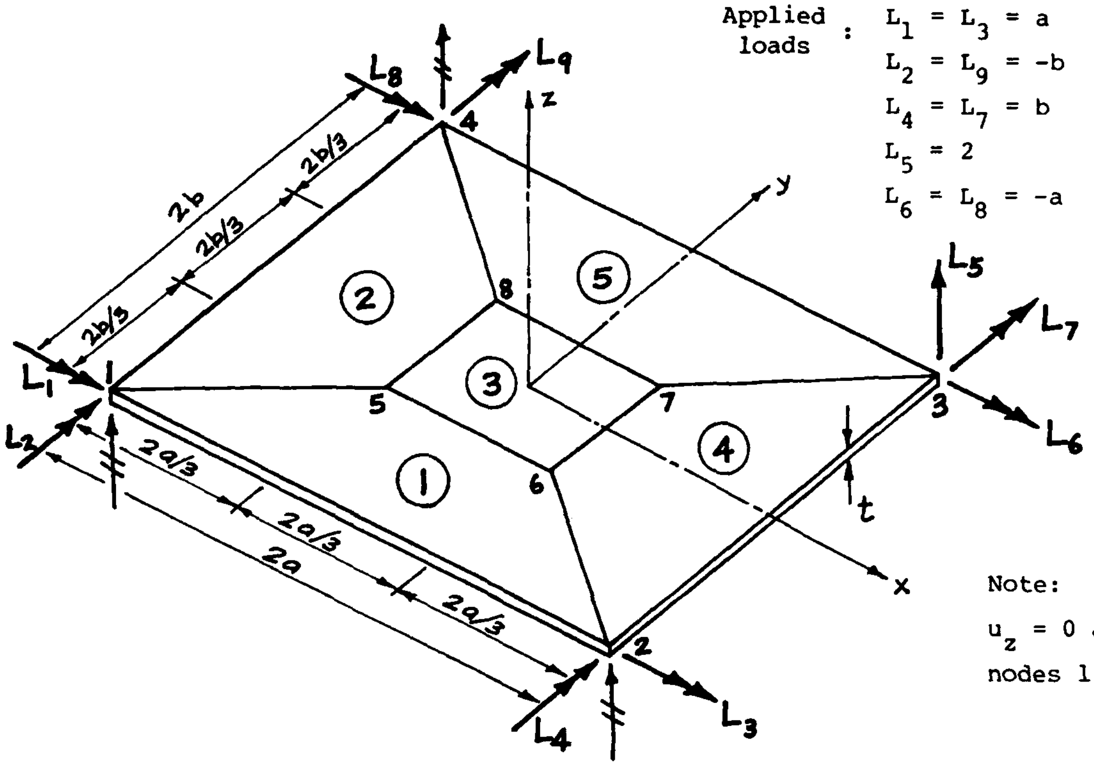



FIGURE 7.2.2-2. CONVERGENCE TEST FOR QDPLT. CLAMPED SQUARE PLATE (AR = 1) WITH CENTRAL POINT LOAD.

### 7.2.3. Patch Test

The patch test, first proposed by B. Irons in 1972, requires a nonconforming element to produce constant stresses/strains when these elements are formed together in a model which has at least one internal node. It thus measures completeness for an assemblage of elements. For a quadrilateral plate bending element such as QDPLT, one possible patch test is this "picture frame" model subjected to the loads shown (Ref. 31):

Applied loads: $L_1 = L_3 = a$
$L_2 = L_9 = -b$
$L_4 = L_7 = b$
$L_5 = 2$
$L_6 = L_8 = -a$




Note:
$u_z = 0$ at
nodes 1, 2, 4

When we used such a model for QDPLT, and assuming:

$a = b = 0.5$, $E = 10^7$ psi, $\nu = 0.3$, $t = 0.1$ in.

We obtained stress results as follows:

Elements 1, 2, 4, 5: $\sigma = 1421$ psi

Element 3: $\sigma = 1138$ psi

QDPLT therefore fails the patch test. The stress values given are the Von Mises effective stresses at the centroid of the element. The Von Mises effective stress was chosen as a measure of stress level because the orientation of the element i-j nodes defines the stress output orientation and therefore stress components are not directly comparable from element to element. This result indicates that an assembly of elements will not produce identical stresses in each element even though the applied loads should produce a uniform stress rate. This is not an unexpected result, however, since slope compatibility is not satisfied at the mutual boundaries of the elements.

In order to determine which type of loading produced the most severe disparity, the combined loading was divided into individual load components as shown in Figure 7.2.3-1. As can be seen, the loading consists of edge moments, $M_x$ and $M_y$, and an out-of-plane force at one corner. The two moment load cases should produce a constant curvature, $R_x$ or $R_y$, resulting in a uniform stress distribution on each surface of the plate. The third loading condition produces a twisting load on the plate and should result in a constant state of stress in all elements. The results indicate that the moment loads produce the largest range of stress values in the elements.

A detailed examination of the displacement solution revealed that the slopes were discontinuous across the element boundaries, therefore the constant curvatures were not correctly predicted. It was interesting to note that the concentrated load at the corner resulted in a fairly uniform stress field, even though this appears to be the most severe loading condition due to the warping of the element. However, the single element test, Figure 7.2-7, does indicate that the element will perform well under this loading as long as moment constraints are not applied (as in Case 8, Figure 7.2-8). Possibly, the reason for the poor performance of the QDPLT element in modeling the constant curvature is related to its poor performance in Cases 6 and 8 of the single element tests, where slope constraints were enforced at the support.

The poor performance of QDPLT in the patch test illustrates the need for the analyst to be aware of the limitations of QDPLT or any other plate bending elements that are being used in a structural analysis. The results should

be checked thoroughly, especially in areas where large stress or strain gradients occur. Also, the fineness of the mesh should be such that the difference in stress between adjacent elements is relatively small. Areas of the mesh that exhibit large stress differences between elements should either be reanalyzed with a finer mesh or results should be assigned a fairly generous error range when the stresses are used to assess the adequacy of the structure.

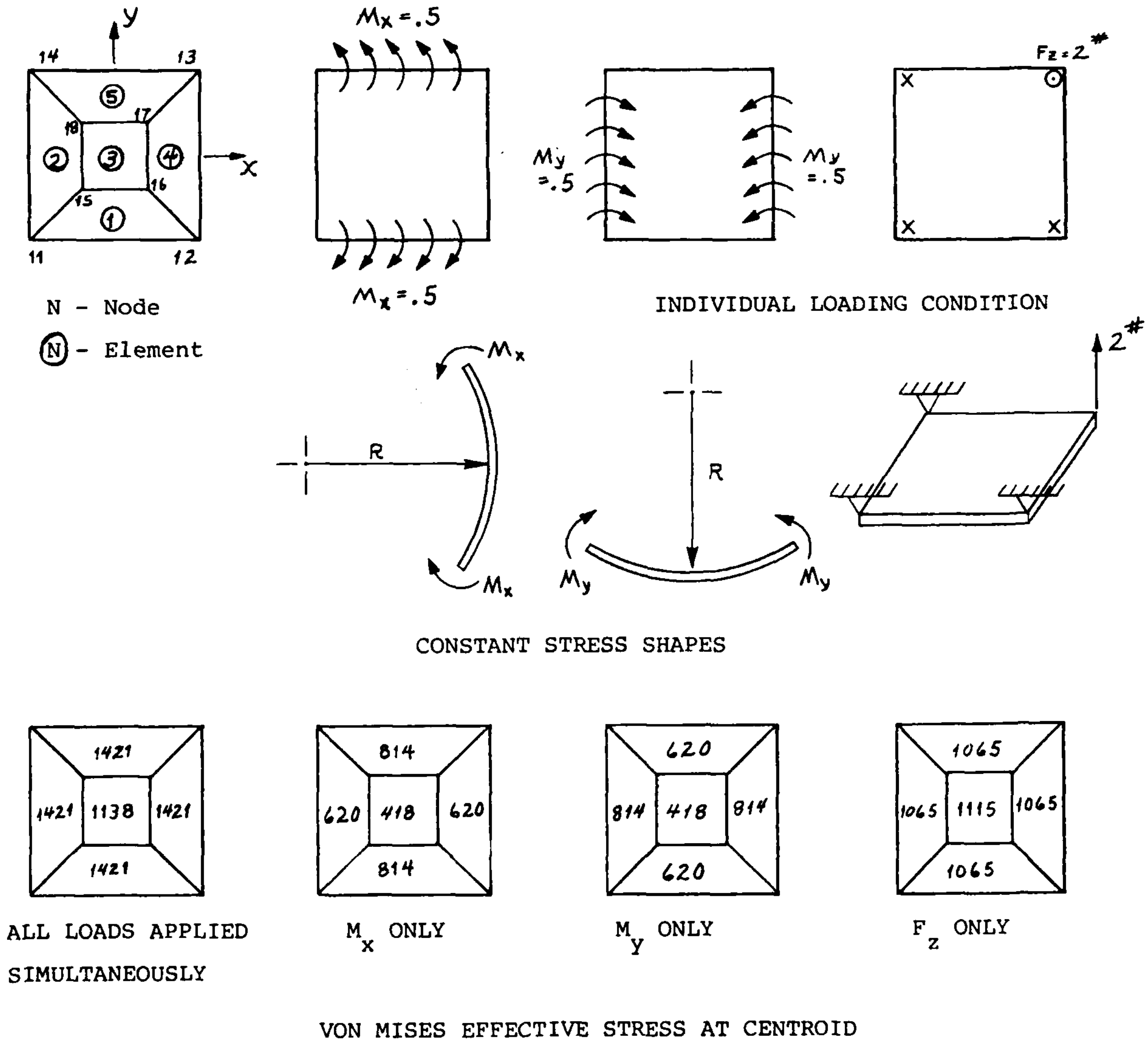



FIGURE 7.2.3-1. QDPLT PATCH TEST RESULTS FOR INDIVIDUAL LOADING CONDITIONS.

### 7.2.4. Element Completeness Test

The completeness requirement for an element is satisfied if the rigid body displacements and constant strain states are represented. A rigid body displacement is the most elementary deformation that an element may undergo. This condition states that there should exist combinations of values of the generalized coordinates $q_i$ that cause all points on the element to experience the same displacement. One such combination should occur for each of the rigid body translations and rotations. In a typical displacement model, the constant term $q_1$ provides for a rigid body displacement.

The constant strain requirement can be stated in similar terms. There should exist combinations of values of the generalized coordinates $q_i$ that cause all points on the element to experience the same strain. As a body is subdivided into smaller and smaller elements, the strains in each infinitesimal element approach constant values. Unless these constant strains are included in the shape function, convergence to the correct solution will not occur. In a typical displacement model, the terms ($q_2x$) and $q_3y$) provide for constant strains $\varepsilon_x$ and $\varepsilon_y$ in an element.

QDPLT is a quadrilateral bending element which is composed of four overlapping basic bending triangles. Upon close examination of the theoretical development of this element in Ref. 7, it was concluded that the terms for both rigid body displacements and constant strain states are included in the displacement model.

### 7.2.5. False Zero Energy Mode Test

If the number of element independent deformation variables plus the number of rigid body modes is less than the number of nodal degrees of freedom in the local system, an element may contain falze zero energy modes (Ref. 31,34). Zero energy modes are always associated with rigid body modes since these give zero elastic energy and zero stress or strain fields. Even when the number of independent deformation variables plus rigid body modes is equal to the number of nodal degrees of freedom, false modes can exist.

False zero energy modes can be found by: numerical techniques directly using the element matrices; an iterative static method of gradually increasing the number of support freedoms (Ref. 31); or, a modal procedure whereby the eigenvalues are computed for an unsupported structure (Ref. 34), as suggested by

Wilson, Taylor, and Doherty (1969). Although a single element may contain false zero energy modes, in many cases experience has shown that these modes do not propagate when combined with other elements to form a structural model, therefore causing no problems. To perform this test rigorously for QDPLT would have meant setting up either: a static method of iteratively searching for false modes by gradually increasing the number of support freedoms---for a variety of supports and loading cases; or an exhaustive eigenvalue interpretation procedure similar to what was done for the NASTRAN solid elements (Subsection 7.7.2.3). Because of the relative lack of importance of this test and our extensive work already previously reported to characterize QDPLT performance in the single element test, convergence test, and patch test, it was decided to forego the false zero energy mode test on QDPLT.

### 7.2.6. Conclusions

Advanced evaluation tests on the QDPLT bending element show it rated fair in performance. In summary, it may be said that QDPLT: performed well in the single element test in six out of eight cases of various loading/support combinations (the exceptions being the twist load cases 6 and 8); converged satisfactorily in the two convergence tests of a simply-supported and clamped square plate under a concentrated center load; and failed the patch test. Its overall performance is inferior to other currently available 4-noded plate elements, such as: MSC/NASTRAN's QUAD4, Lockheed-California NASTRAN's LORA stress-based plate bending element, and T. H. H. Pian's stress-based element with nine independent force variables (Ref. 33).

## 7.3. The TRSHL Element

TRSHL is a triangular shallow shell element implemented into Level 16.0 of COSMIC/NASTRAN in 1976. When TRSHL was originally selected for advanced evaluation, it seemed a logical choice because it is a higher-order shell element recently incorporated into the COSMIC/NASTRAN element library. In this subsection, a description of TRSHL documentation, theory, status, and efficiency will be given. Most, or all of this background information is unavailable to the average NASTRAN user, and it is included to illustrate the implementation history and current status of a typical element in the NASTRAN library.

### 7.3.1. Background

TRSHL was developed and implemented into COSMIC/NASTRAN by Dr. R. Narayanaswami in 1974-1976 (References 37,38) while he was at Old Dominion University, Norfolk, Virginia. The element models curved thin shell behavior by approximating: the membrane behavior using the linear strain TRIM6 element, the bending behavior using the quintic transverse displacement TRPLT1 element, and membrane-bending coupling using Novozhilov shallow-shell theory. This 6-noded element has 30 degrees of freedom; at each node there are five degrees of freedom consisting of three translations and two rotations. The element is designed for use with the statics, normal modes, and buckling rigid formats of NASTRAN.

In our early investigations, two interesting things were discovered about TRSHL. First, the documentation of TRSHL in the NASTRAN manuals was poorly written, short, and incomplete. Narayanaswami's two Old Dominion University reports cited in the NASTRAN Theoretical Manual were unavailable through the usual NASA or NTIS publication sources, even though the research was funded by NASA/Langley. (We were able to obtain Reference 37 by writing directly to Old Dominion University). Second, although TRSHL still exists in the COSMIC/NASTRAN Level 17.5 element library, its use appears to be limited and has been abandoned in both the latest versions of NASTRAN offered by the MacNeal-Schwendler Corporation (MSC) and Universal Analytics, Inc. (UAI).

The history of the development and implementation of the TRSHL element illustrates one of the weaknesses of the piecemeal development of a large general purpose program such as NASTRAN. The use of a number of different subcontractors to provide finite elements or solution algorithms often leads

to a situation where the developer is not available to completely debug the program modification in its final form. Furthermore, the completeness and accuracy of the documentation often vary significantly. Additionally, the developer is usually not available to provide user support for this part of the program. This latter criticism is often cited (for example Ref. 22) as an important evaluation point in comparing computer programs.

Universal Analytics, Inc. was contacted to inquire why TRSHL is not included in UAI/NASTRAN anymore. In response, Mr. M. J. Morgan of UAI informed us that UAI found TRSHL did have poor performance and reputation, and UAI had decided instead to implement its own QUAD3 and QUAD8 elements for shell analysis. He referred us to Dr. A. B. Potvin of Exxon Production Research Company (Houston, Texas) for more TRSHL evaluation results. Dr. Potvin told us that he discovered several minor coding errors in the TRSHL implementation into Level 16.0 of COSMIC/NASTRAN, and after correcting these himself, he found its performance and convergence to be relatively poor in the Scordelis-Lo problem (Ref. 39). Potvin's TRSHL data are included in Figure 7.3-2.

Based upon these inquiries, the following conclusions have been drawn concerning TRSHL: The documentation in NASTRAN is inadequate; its use is apparently quite limited; there is little technical support available from its developer; its performance is inferior (as can be seen from Figure 7.3-2) to other currently available shell elements, and its coding implementation into COSMIC/NASTRAN was apparently not thoroughly tested.

### 7.3.2. Theory

The TRSHL element uses only displacements and rotations as nodal degrees of freedom (Figure 7.3-1). The displacement functions for u, v, and w are chosen as follows:

$$u = q_1 + q_2x + q_3y + q_4x^2 + q_5xy + q_6y^2$$

$$v = q_7 + q_8x + q_9y + q_{10}y^2 + q_{11}xy + q_{12}y^2$$

$$w = q_{13} + q_{14}x + q_{15}y + q_{16}x^2 + q_{17}xy + q_{18}y^2 + q_{19}x^3 + q_{20}x^2y + q_{21}xy^2 + q_{22}y^3 + q_{23}x^4 + q_{24}x^3y + q_{25}x^2y^2 + q_{26}xy^3 + q_{27}y^4 + q_{28}x^5 + q_{29}x^4y + q_{30}x^3y^2 + q_{31}x^2y^3 + q_{32}xy^4 + q_{33}y^5$$

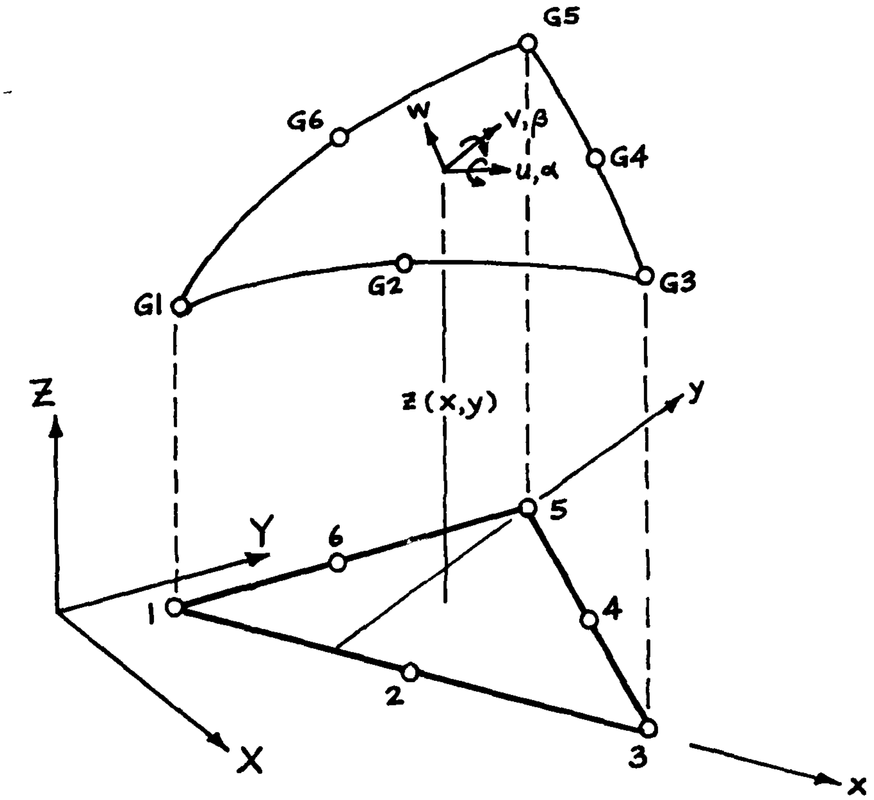



FIGURE 7.3-1. TRSHL SHELL ELEMENT GEOMETRY AND COORDINATE SYSTEMS.

The element includes transverse shear flexibility in the stiffness formulation. Use of quadratic polynomials for the shell surface geometry implies constant curvatures, thereby agreeing with the approximations of shallow shell theory.

7.3.3. Evaluation Results

Although Narayanaswami presented two numerical examples (spherical cap, Scordelis-Lo cylindrical shell), here we will comment on only the latter. Table 7.3-1 shows TRSHL results for the Scordelis-Lo problem, and the trends of four predicted displacements and three stress resultants versus mesh size. Note that even for the fine 6 x 6 mesh, discrepancies are quite pronounced for four of the seven results: $V_B$, $W_C$, $M_{yy_C}$, $M_{xx_C}$.

Narayanaswami (Ref. 37) did not provide a complete explanation for the difference between the calculated values and the exact solution. If one examines only the free edge midpoint vertical displacement (Figure 7.3-2), TRSHL fared somewhat better, though it is still inferior to Cowper, et. al. (1970) and many other available commercial shell elements. Potvin's independent check of TRSHL results (Ref. 39) is also shown: it is noted that there is a considerable difference between Potvin's results and those reported by Narayanaswami. It is also interesting to note that, although TRSHL is described in Reference 7, it was omitted in a table which lists available COSMIC/NASTRAN finite elements in Reference 11.

7.3.4. Conclusions

Based on the results of this investigation, it is concluded that TRSHL element quality and efficiency rate considerably below those of currently available shell elements in competing software. A low (and declining) level of its use in the future is predicted, leading to its probable ultimate demise in the COSMIC/NASTRAN element library.

TABLE 7.3.1. TRSHL RESULTS FOR SCORDELIS-LO CYLINDRICAL SHELL ROOF PROBLEM

| Finite Element Grids | $10u_A$ (in.) | $w_B$ (in.) | $10v_B$ (in.) | $10w_C$ (in.) | $10^{-3}N_{xxB}$ (lb./in.) | $10^{-3}M_{yyC}$ (lb. in./in.) | $10^{-2}M_{xxC}$ (lb. in./in.) |
|---|---|---|---|---|---|---|---|
| 1 x 1 | -0.45168 | -0.29100 | -2.48424 | -4.0700 | 2.4659 | 0.7685 | 2.8520 |
| 2 x 2 | -0.7812 | -1.2516 | -4.77312 | -2.1344 | 4.2801 | -0.9395 | -0.8896 |
| 3 x 3 | -1.09590 | -2.49876 | -7.12872 | -1.3606 | 5.4948 | -2.0283 | -1.1136 |
| 4 x 4 | -1.2939 | -3.4332 | -8.57580 | 2.2224 | 6.0277 | -2.3828 | -1.7912 |
| 2 x 4 | -0.9041 | -2.2815 | -6.15 | 0.888 | 5.1312 | -1.415 | -2.0196 |
| 3 x 6 | -1.1244 | -3.6227 | -8.5968 | 3.1031 | 6.1862 | -1.9414 | -1.8912 |
| 4 x 8 | -1.3845 | -4.1526 | -9.5295 | 3.9238 | 6.4839 | -2.0459 | -1.6724 |
| 5 x 5 | -1.4160 | -3.88152 | -9.29000 | 2.8182 | 6.3279 | -2.3538 | -1.9770 |
| 6 x 6 | -1.4733 | -4.09176 | -9.76992 | 3.0900 | 6.444 | -2.3242 | -2.0638 |
| Exact (Scordelis-Lo) | -1.51325 | -4.09916 | -8.76147 | 5.2494 | 6.4124 | -2.0562 | -0.9272 |

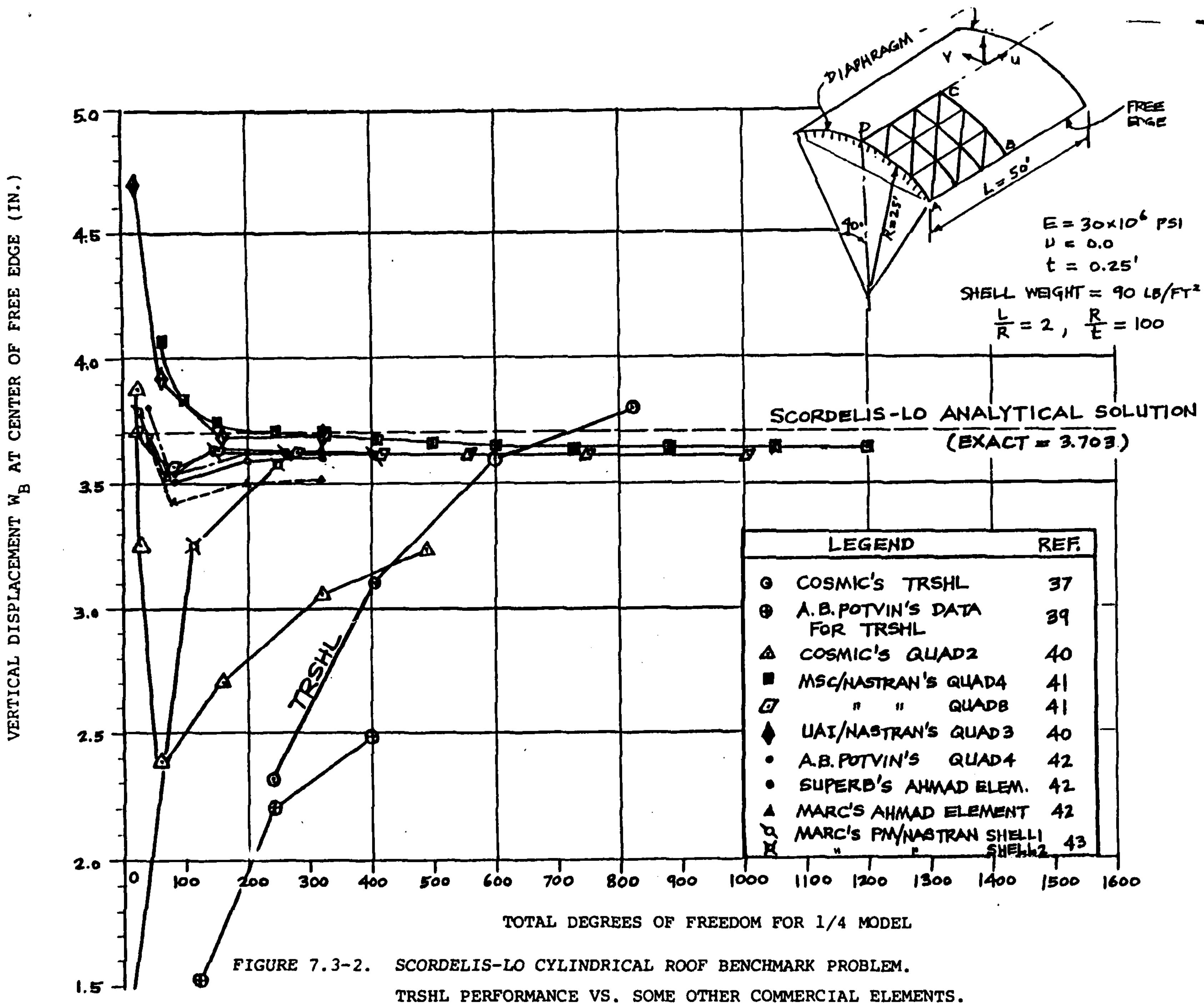



FIGURE 7.3-2. SCORDELIS-LO CYLINDRICAL ROOF BENCHMARK PROBLEM.
TRSHL PERFORMANCE VS. SOME OTHER COMMERCIAL ELEMENTS.

## 7.4. The TRIM6 and TRPLT1 Elements

Closely related to TRSHL are the linear strain triangular membrane element TRIM6 and the higher order triangular bending element TRPLT1. This section gives a short summary of the element theory and efficiency for both elements. Evaluation results are already well documented in References 7 and 38. TRIM6, TRPLT1, and TRSHL were added to Level 16.0 of COSMIC/NASTRAN by Dr. Narayanaswami in December 1976.

### 7.4.1. Theory

The linear strain triangular membrane element TRIM6 is based on J. H. Argyris' derivation in 1965. The element has six nodes, three at the vertices and three at the midpoints of the sides. The element uses a quadratic displacement field.

$$u = q_1 + q_2x + q_3y + q_4x^2 + q_5xy + q_6y^2$$

$$v = q_7 + q_8x + q_9y + q_{10}x^2 + q_{11}xy + q_{12}y^2$$

The thickness of the element, as well as the temperature distribution within the element, can have bilinear variation. TRIM6 may be used for the statics and normal modes rigid formats, but cannot be used for differential stiffness and piecewise linear analyses. Element stresses are computed at the three vertices and at the centroid.

TRPLT1 was developed by Narayanaswami in 1974 as a modification of Cowper et. al. (1968). Like TRIM6 and TRSHL, TRPLT1 has six nodes, three at the vertices and three at the midpoints of the sides. A quintic displacement field is chosen for the transverse displacement w (see Subsection 7.3.2). The element is nonconforming, meaning it has no slope continuity for two adjacent elements with a common edge. Each node has three degrees of freedom: two rotations and the transverse displacement w. Transverse shear flexibility is included in the stiffness formulation. The element thickness can have bilinear variations. TRPLT1 may be used in the statics and normal modes rigid formats. The element internal forces are recovered at the three vertices and at the centroid.

### 7.4.2. Evaluation Results

The performances of TRIM6 and TRPLT1 are already adequately described in References 7 and 38. We decided not to repeat these exercises, and only report

here some representative test problems which demonstrate their efficiency and convergence characteristics.

- Example 1. Cantilevered beam using membrane elements.

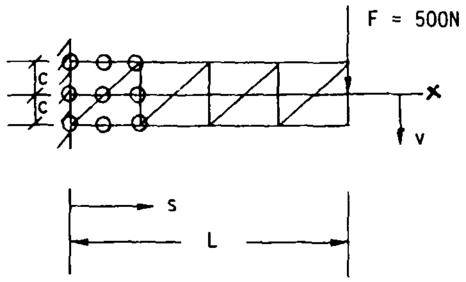
  Figure 7.4-1 shows the excellent correlation of TRIM6 with theory and its superiority over the QDMEM1 isoparametric membrane element and the conventional quadrilateral membrane element QDMEM. This example demonstrates the improved accuracy due to use of a quadratic displacement polynomial, and appears in both References 7 and 38.

- Example 2. Rectangular plate bending using TRPLT1.

  Figure 7.4-2 gives a typical TRPLT1 deflection convergence study for a rectangular plate, simply supported at the four edges and with a central concentrated load. Note that TRPLT1 converges monotonically from above the theoretical value, a behavior which is typical of nonconforming elements and suggesting need for a patch test. This example was for a (b/a) ratio of 2.

### 7.4.3. Conclusions

A careful review of all the convergence efficiency, and demonstration examples in References 7 and 38 reveals that the TRIM6 and TRPLT1 elements performed quite well for a variety of conditions and load cases - including statics, thermal loads, vibration, and buckling problems.




TRIM6 modeling scheme

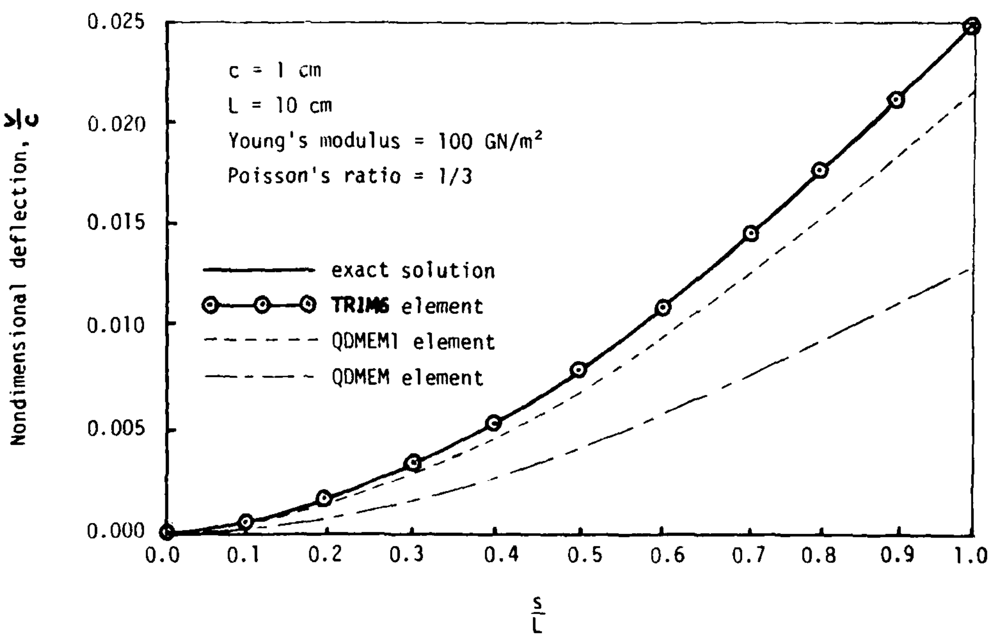



**FIGURE 7.4-1. DEFLECTION OF CANTILEVER BEAM IDEALIZED BY QDMEM, QDMEM1, AND TRIM6 ELEMENTS.**

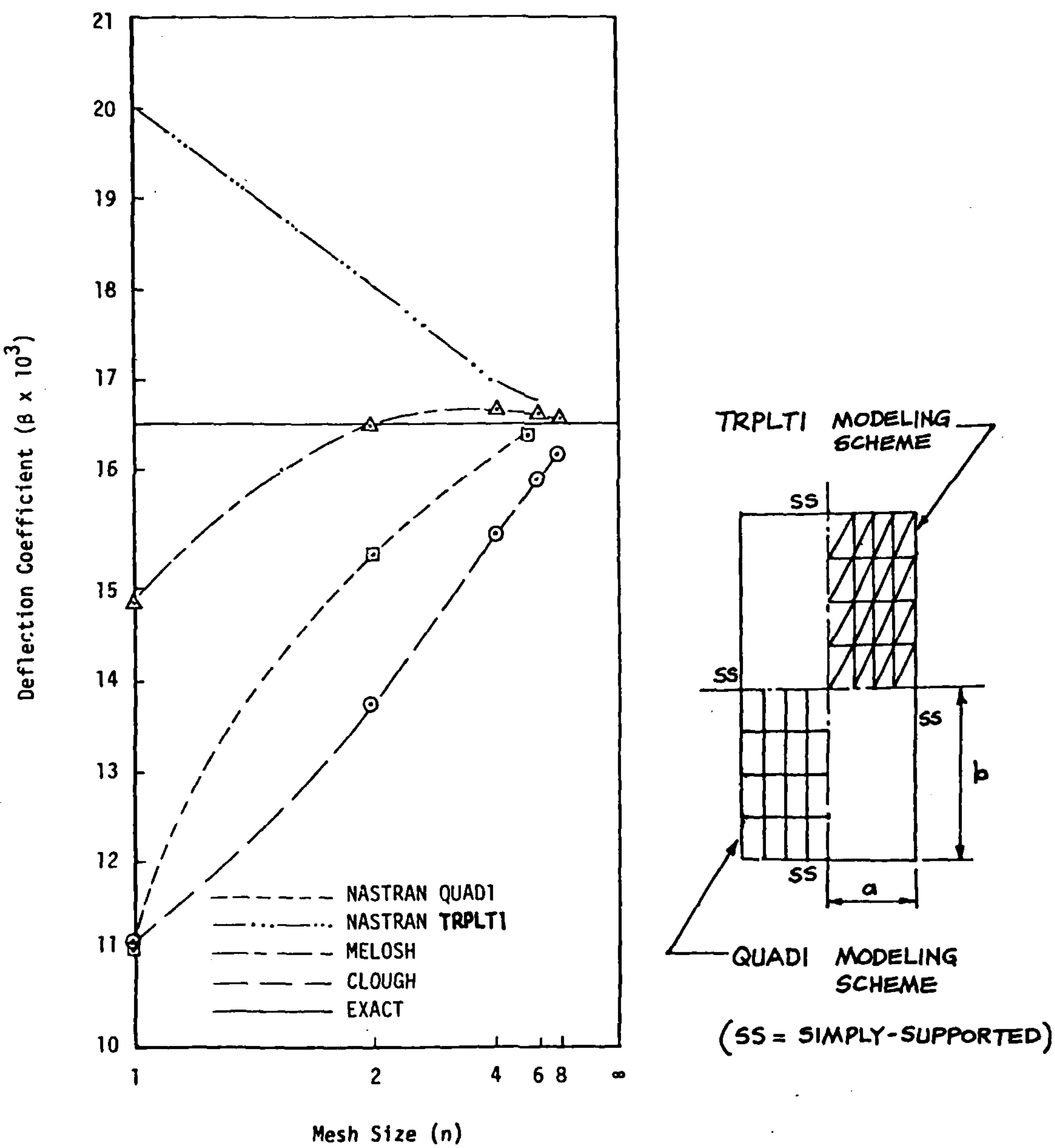


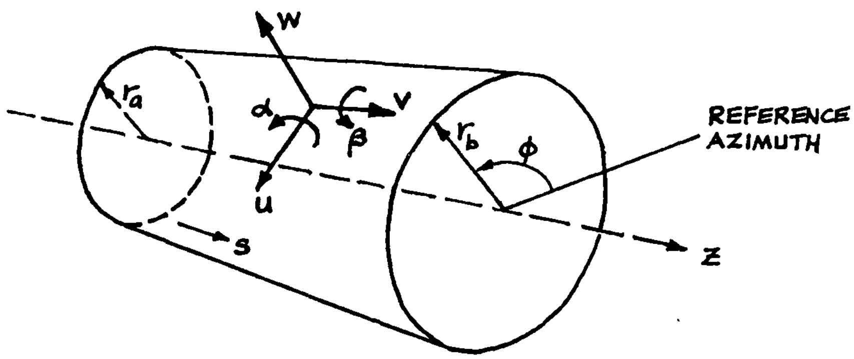

FIGURE 7.4-2. CENTRAL DEFLECTION OF RECTANGULAR PLATE.

## 7.5. The CONEAX Element

CONEAX is a conical shell element which can have either axisymmetric or asymmetric loading. This section briefly explains CONEAX theory and our advanced evaluation results using an edge-loaded cylinder to assess its accuracy.

### 7.5.1. Theory

The CONEAX element coordinate geometry and 5 degrees of freedom are shown below.




The displacements u and v are assumed to be linear functions of the meridional position s, while w is assumed to be a cubic. For each harmonic order $q_n$, the harmonic components of deflection are (Ref. 7):

$$u_n(s) = q_{1n} + q_{2n}s$$

$$v_n(s) = q_{3n} + q_{4n}s$$

$$w_n(s) = q_{5n} + q_{6n}s + q_{7n}s^2 + q_{8n}s^3 \quad .$$

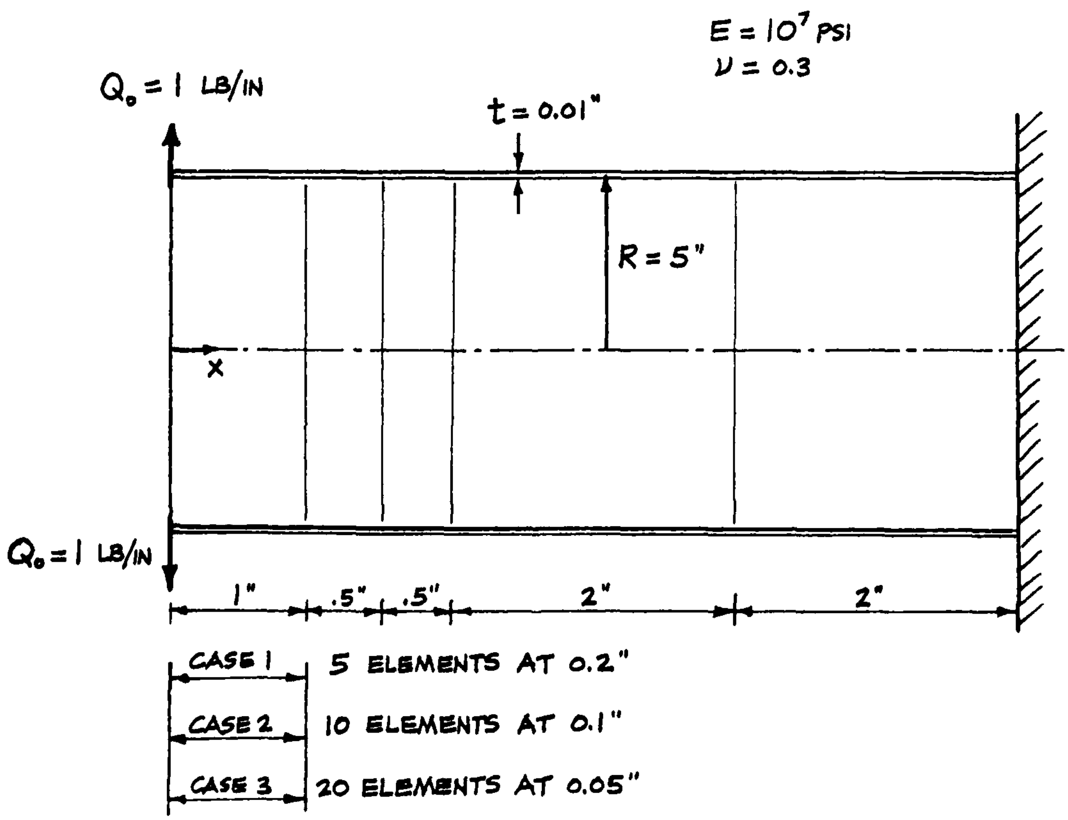
Therefore, CONEAX is similar in derivation to the conical shell element of Grafton and Strome (Ref. 44). In a conical assemblage the structure modeled by this conical element is permitted to deform only approximately but is loaded in a manner consistent with those deformations. The philosophy is that many small elements, each

deforming in a simple way, should provide a good approximation to the actual structure. In any region of the structure in which the character of the deformation varies rapidly, it is necessary to use a fairly large number of such elements. Clearly, this is the case in edge bending calculations (Ref. 45).

Properties of CONEAX are assumed axisymmetric. However, the loads and deflections need not be axisymmetric; they are expanded in Fourier series with respect to the azimuth coordinate. An unusual feature of CONEAX is that it includes transverse shear flexibility. The rotations $\alpha$ and $\beta$ are independent motions because of the transverse shear flexibility. Rotation about the normal to the surface is not included, since such rotation can be adequately represented by the gradients of u and v. The element cannot be combined with other NASTRAN structural elements in the solution of problems.

7.5.2. Advanced Evaluation

To evaluate the convergence and accuracy of CONEAX, the radial-loaded cylinder example used in Grafton and Strome's paper (Ref. 44) and by many subsequent researchers was selected as the benchmark. This example features rapidly varying deflections and stress resultants.



Theoretical predictions for the radial deflection and meridional moment can be obtained using Hetenyi (Ref. 46):

Radial deflection: $$w = \frac{Q_o}{2\beta^3 D} e^{-\beta x} \cos \beta x$$

Meridional moment: $$M = \frac{Q_o}{\beta} e^{-\beta x} \sin \beta x$$

where: ($\nu$=.3)

$$\beta = \frac{1.285}{\sqrt{Rt}} = \frac{1.285}{\sqrt{5(.01)}} = 5.746694$$

$$D = \frac{Et^3}{12(1-\nu^2)} = \frac{10^7(.01)^3}{12(1-.3^2)} = 0.915751$$

Computed results using CONEAX were remarkably close to theory, even for Case 1 (5 elements). All three models produced consistently better correlation with theory for radial deflection and meridional moment, as compared to Grafton and Strome (Ref. 44). Results are listed below and plotted in Figures 7.5-1 and 7.5-2.

.Radial Deflection:

| | | CONEAX RESULTS ($10^{-3}$ in.) | | |
|---|---|---|---|---|
| x (in) | THEORY ($10^{-3}$ in) | Case 3 (20 elements) | Case 2 (10 elements) | Case 1 (5 elements) |
| 0.00 | 2.8769 | 2.8723 | 2.8623 | 2.8152 |
| 0.05 | 2.0699 | 2.066 | | |
| 0.10 | 1.3919 | 1.356 | 1.3527 | |
| 0.15 | 0.79086 | 0.7888 | | |
| 0.20 | 0.37291 | 0.3722 | 0.37259 | 0.3708 |
| 0.25 | 0.091453 | 0.09145 | | |
| 0.30 | -0.07831 | -0.07775 | -0.07555 | |
| 0.35 | -0.16416 | -0.16326 | | |
| 0.40 | -0.19215 | -0.1911 | -0.1892 | -0.1811 |
| 0.45 | -0.18410 | -0.1831 | | |
| 0.50 | -0.15676 | -0.1558 | -0.1547 | |
| 0.55 | -0.12195 | -0.1212 | | |
| 0.60 | -0.08725 | -0.0867 | -0.0862 | -0.0831 |
| 0.65 | -0.05690 | -0.0565 | | |
| 0.70 | -0.03278 | -0.03261 | -0.0325 | |

.Meridional Moment: ($\bar{x}$ is at element centroid)

| | 20 ELEMENTS | | | 10 ELEMENTS | | | 5 ELEMENTS | | |
|---|---|---|---|---|---|---|---|---|---|
| | Case 3 ($10^3$ in-lbs) | | | Case 2 ($10^3$ in-lbs) | | | Case 1 ($10^3$ inches) | | |
| Element | $\bar{x}$ (in.) | Theory | CONEAX | $\bar{x}$ (in.) | Theory | CONEAX | $\bar{x}$ (in.) | Theory | CONEA |
| 1 | 0.025 | 21.58 | 20.39 | | | | | | |
| | | | | 0.05 | 36.99 | 33.23 | | | |
| 2 | 0.075 | 47.24 | 46.36 | | | | | | |
| | | | | | | | 0.10 | 53.24 | 43.04 |
| 3 | 0.125 | 55.83 | 55.23 | | | | | | |
| | | | | 0.15 | 55.78 | 54.04 | | | |
| 4 | 0.175 | 53.75 | 53.36 | | | | | | |
| 5 | 0.225 | 45.92 | 45.71 | | | | | | |
| | | | | 0.25 | 40.99 | 40.52 | | | |
| 6 | 0.275 | 35.82 | 35.74 | | | | | | |
| | | | | | | | 0.30 | 30.67 | 30.21 |
| 7 | 0.325 | 25.71 | 25.70 | | | | | | |
| | | | | 0.35 | 21.06 | 21.18 | | | |
| 8 | 0.375 | 16.82 | 10.86 | | | | | | |
| 9 | 0.425 | 9.379 | 9.805 | | | | | | |
| | | | | 0.45 | 6.912 | 7.192 | | | |
| 10 | 0.475 | 4.545 | 4.619 | | | | | | |
| | | | | | | | 0.50 | 2.606 | 3.56 |
| 11 | 0.525 | 1.058 | 1.129 | | | | | | |
| | | | | 0.55 | -0.1408 | 0.0953 | | | |
| 12 | 0.575 | -1.035 | -0.9753 | | | | | | |
| 13 | 0.625 | -2.085 | -2.039 | | | | | | |
| | | | | 0.65 | -2.323 | -2.185 | | | |
| 14 | 0.675 | -2.418 | -2.386 | | | | | | |
| | | | | | | | 0.70 | -2.403 | -2.05 |
| | | | | 0.75 | -2.151 | -2.097 | | | |

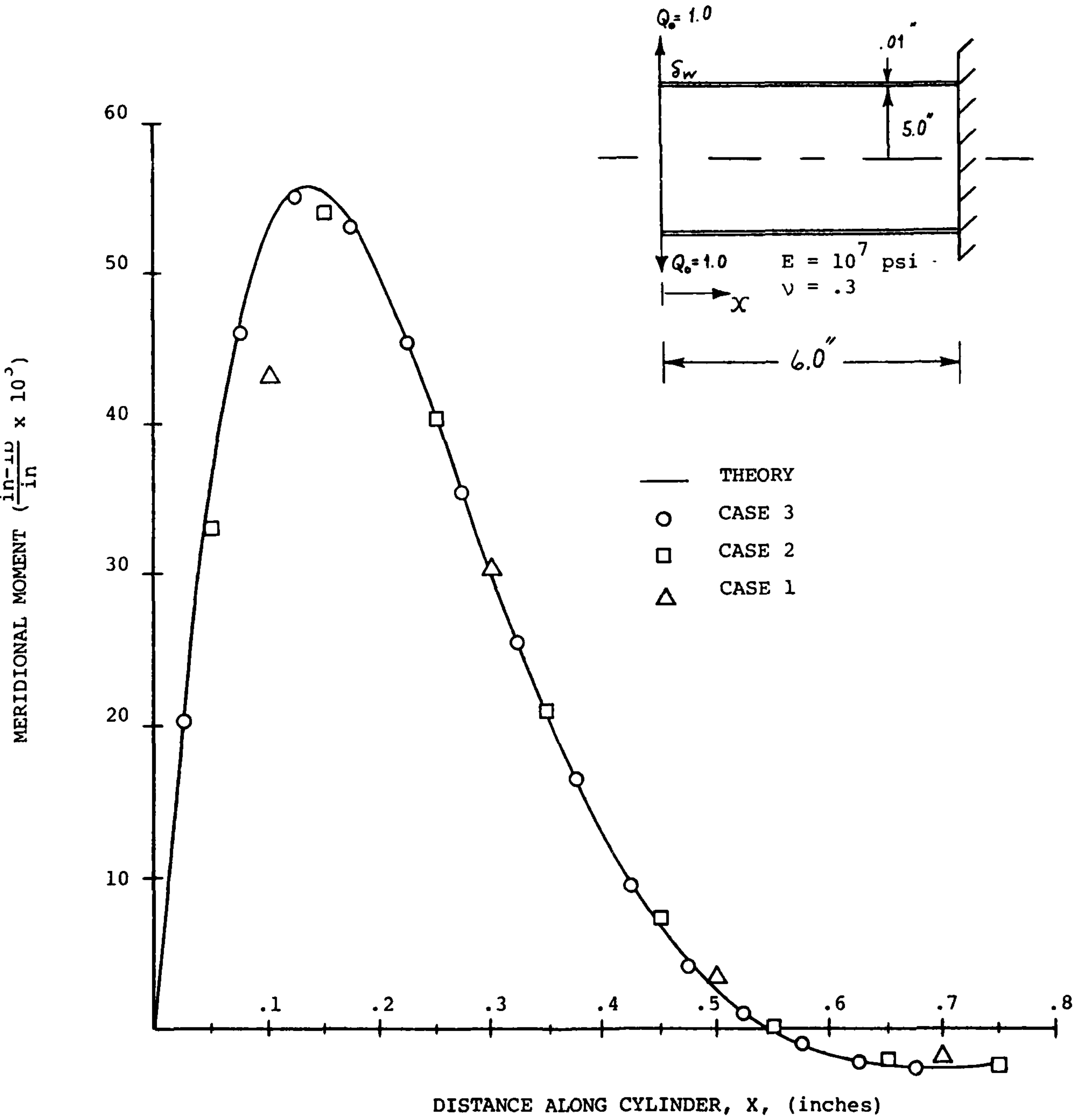



FIGURE 7.5-1. RADIAL DEFLECTION - CONEAX VS. THEORY.

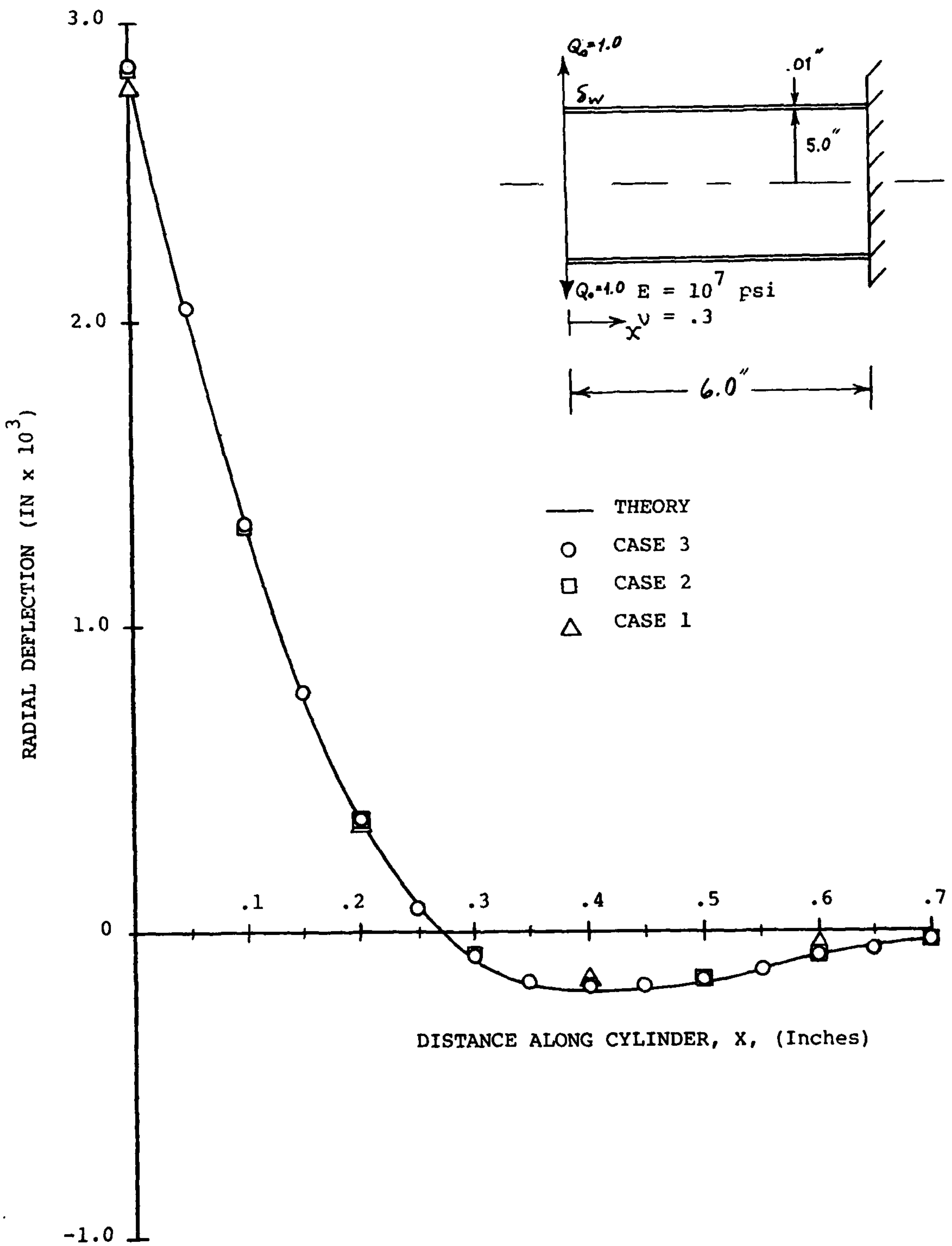



FIGURE 7.5-2. MERIDIONAL MOMENT - CONEAX VS. THEORY.

### 7.5.3. Other CONEAX Uses

In addition to handling axisymmetric loadings, CONEAX is versatile in its use and can be applied to a variety of other loading conditions:

- Unsymmetric loading - using standard Fourier expansion techniques
- Line loads - uniformly distributed or harmonic coefficients
- Concentrated loads - at specified azimuth positions
- Pressure loads - normal to surface
- Gravity loads
- Thermal loads
- Enforced strains
- Enforced displacements
- Branched shells

### 7.5.4. Conclusions

Our investigations indicate that CONEAX is a good conical shell element, with good convergence properties and exceptional versatility. Only the axisymmetric mechanical load capability was evaluated in this study. It has been reported (Ref. 53) that the thermal loading capability that exists may contain errors and is not recommended for use.

## 7.6. The TRAPRG and TRAPAX Elements

TRAPRG is one of the four solid-of-revolution elements offered in COSMIC/NASTRAN. It is a four-noded axisymmetric trapezoidal ring element with axisymmetric loading capability. The other three are: TRIARG, same as TRAPRG except it is triangular; TRAPAX and TRIAAX, axisymmetric trapezoidal and triangular ring elements with nonaxisymmetric loading capability. The benchmark problem selected to test TRAPRG and TRAPAX performance is a thick-walled cylinder under internal pressure, one used by many past researchers, and a good test to measure their capability to represent simple radial dilatation behavior.

### 7.6.1. Theory

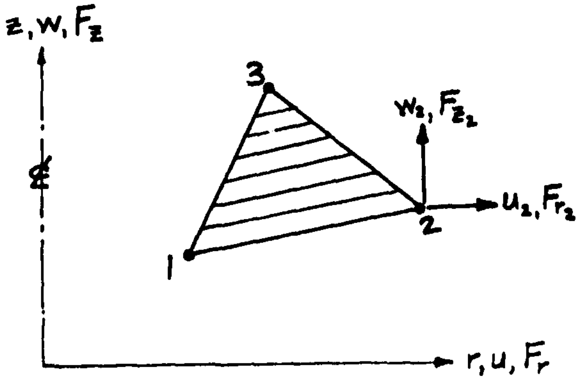
According to the Theoretical Manual (Ref. 7), the formulation of the theory behind TRAPRG is "mathematically consistent" with that in Mallet and Jordan's MAGIC (1969) and MAGIC II (1971) codes developed at the Air Force Flight Dynamics Laboratory. The two original finite element papers on the subject of structural analysis of axisymmetric solids were by Clough and Rashid (1965) and Wilson (1965). There was apparently (Ref. 7) an extension of the Wilson formulation to include orthotropic material properties for TRAPRG in COSMIC/NASTRAN. The TRAPRG theoretical derivations are quite standard for solid-of-revolution elements. The 31-page documentation in Reference 7 is adequate in describing: coordinate notations; displacement functions (Fourier harmonics in circumferential direction); derivation of the element stiffness matrix and mass matrix; treatment of orthotropic behavior; development of non-axisymmetric load vectors (including thermal loads); prestrain, pressure and thermal load vectors; displacement and stress recovery. The TRAPRG element and the other axisymmetric elements are therefore ten to fifteen years old. They represent fairly typical solid-of-revolution elements available in all general purpose codes and documented in every finite element text, and have been incorporated in NASTRAN from the very beginning of the code.

The basic displacement assumption for TRIARG is:

$$u = q_1 + q_2 r + q_3 z$$

(6 terms)

$$w = q_4 + q_5 r + q_6 z$$



But the generalized harmonic displacements for TRAPRG are:

$$u_n = q_{1n} + q_{2n}r + q_{3n}z + q_{4n}rz$$

(8 terms)

$$w_n = q_{5n} + q_{6n}r + q_{7n}z + q_{8n}rz$$

Stresses for TRAPRG are "evaluated at the four nodal regions as well as at a fifth region which corresponds to a point that is obtained by averaging the coordinates of the four nodal points." (Currently, many other codes already offer more accurate axisymmetric elements with mid-side nodes.)

Any point within a solid-of-revolution element can be located by specifying a radial coordinate (r), an axial coordinate (z), and a circumferential position ($\phi$). In TRAPRG and TRIARG, the point "i" displacements $U_{ri}$ and $U_{zi}$ are constants with respect to $\phi$. In TRAPAX and TRIAAX, the displacements $U_{ri}$, $U_{zi}$, and $U_{pi}$, as well as the forces $F_{ri}$, $F_{zi}$, and $F_{\phi i}$ at each point i are assumed to be sinusoidal functions of the circumferential location $\phi$. Directly applied nonaxisymmetric loads, pressure loads, gravity loads, and thermal loads are applied to the TRAPAX and TRIAXX elements with the same method as in the conical shell element CONEAX.

Two shortcomings are noted in the documentation of solid-of-revolution elements in NASTRAN. First, there is no explanation or justification for the trapezoidal requirement, that is, two opposite sides of TRAPRG must be parallel. This strict trapezoidal requirement exists only in COSMIC/NASTRAN, and is extremely restrictive on the analyst's choice of mesh to the point of inconvenience. Second, no mention is made of the relationship or similarity of the NASTRAN solid-of-revolution elements to those in the SAAS code. The SAAS II and III codes (and the asymmetric ASAAS code) also utilize the Wilson element, handle orthotropic behavior, and have been extensively used in the aerospace industry for ten years. Moreover, SAAS offers the user a choice of plane strain, plane stress, or axisymmetric analyses, a choice not found in NASTRAN. SAAS has an excellent reputation as a versatile and cheap code to run. It seems to the authors that before and during implementation of the NASTRAN solid-of-revolution elements, there should have been several test cases made to compare NASTRAN efficiency with SAAS III, and to possibly incorporate SAAS numerical algorithms and its powerful Laplacian mesh generation capability. The fact is that an user who has SAAS in-house will probably never use NASTRAN for axisymmetric analysis of solids.

## 7.6.2. Advanced Evaluation

The benchmark problem chosen to evaluate TRAPRG efficiency and convergence characteristics is a thick-walled cylinder (plane strain) under internal pressure. This problem was selected because its elasticity solution is well-known, and it has been used by past researchers, including Clough and Rashid (1965) and Wilson (1965). The theoretical equations for radial and hoop stresses and for radial displacement are (Ref. 47):

$$\sigma_r = \frac{a^2 p}{b^2 - a^2} \left(1 - \frac{b^2}{r^2}\right)$$

$$\sigma_\Theta = \frac{a^2 p}{b^2 - a^2} \left(1 + \frac{b^2}{r^2}\right)$$

$$U_r = \frac{1}{E} \left[ \frac{a^2 P_i}{b^2 - a^2} \left(r + \frac{b^2}{r}\right) (1 - \nu^2) - \frac{a^2 P_i}{b^2 - a^2} \left(r - \frac{b^2}{r^2}\right) (\nu + \nu^2) \right]$$

These equations show that both stresses are maximum at the inner surface where r is minimum, $\sigma_r$ is always a compressive stress and smaller than $\sigma_\Theta$, and $\sigma_\Theta$ is a tensile stress which is maximum at the inner surface.

For assumed values in the test case of $E = 10^7$ psi, $\nu = 0.3$, a = 5 in., b = 10 in., p = 1000, the above expressions become:

| | @ r = 5 in. | @ r = 10 in. |
|---|---|---|
| $\sigma_r = 333.3 - \dfrac{3.333 \times 10^4}{r^2}$ | -1000 psi | 0 psi |
| $\sigma_\Theta = 333.3 + \dfrac{3.333 \times 10^4}{r^2}$ | 1667 psi | 666.7 psi |
| $u_r = (1.733 \times 10^{-5}) r + \dfrac{(4.33 \times 10^{-3})}{r}$ | $9.53 \times 10^{-4}$ in. | $6.07 \times 10^{-4}$ in. |

Since there is no input option to apply pressure on the element face for TRAPRG in COSMIC/NASTRAN, we had to calculate and input equivalent nodal forces. TRAPRG was evaluated using three meshes: 2, 5, and 10 elements through the thickness. Results are compared with theory in Figures 7.6-1 and 7.6-2. The correlation is good.

TRAPAX was also evaluated using the same three meshes, and the excellent pressurized thick-walled cylinder results are plotted in Figures 7.6-3 and 7.6-4. Unlike TRAPRG, TRAPAX does have an input option for pressure directly applied to an element face. We were primarily interested in verifying the accuracy and correctness of coding of TRAPAX. The consistency of results between TRAPRG and TRAPAX verified this coding. Harmonic superposition used to handle non-axisymmetric loads was not evaluated in this study, since this procedure is fairly standard in all the well-known codes. Another reason TRAPAX was not evaluated for nonaxisymmetric loadings was the difficulty of formulating a pathological test case for such loadings.

### 7.6.3. Conclusions

For the "uniform-dilatation" thick-walled cylinder benchmark, our evaluation results showed both TRAPRG and TRAPAX to perform well even for a crude 2-element model. The only drawbacks are the TRAPRG trapezoidal modeling restriction, lack of a direct pressure load input capability for TRAPRG, and the absence of one element in NASTRAN to handle plane strain, plane stress, or axisymmetric analyses.

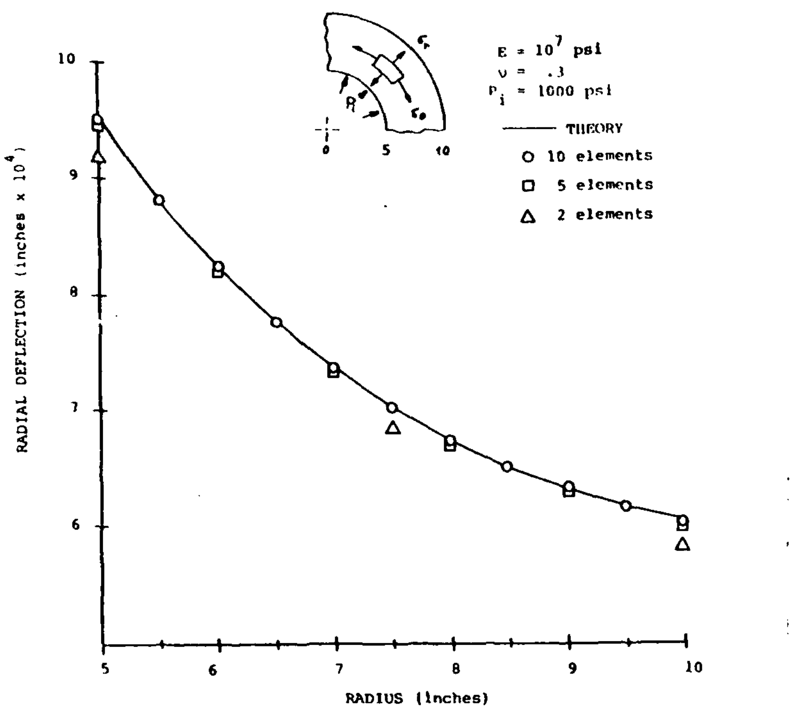



FIGURE 7.6-1. RADIAL DEFLECTIONS OF THICK-WALLED CYLINDER - TRAPRG VS. THEORY.

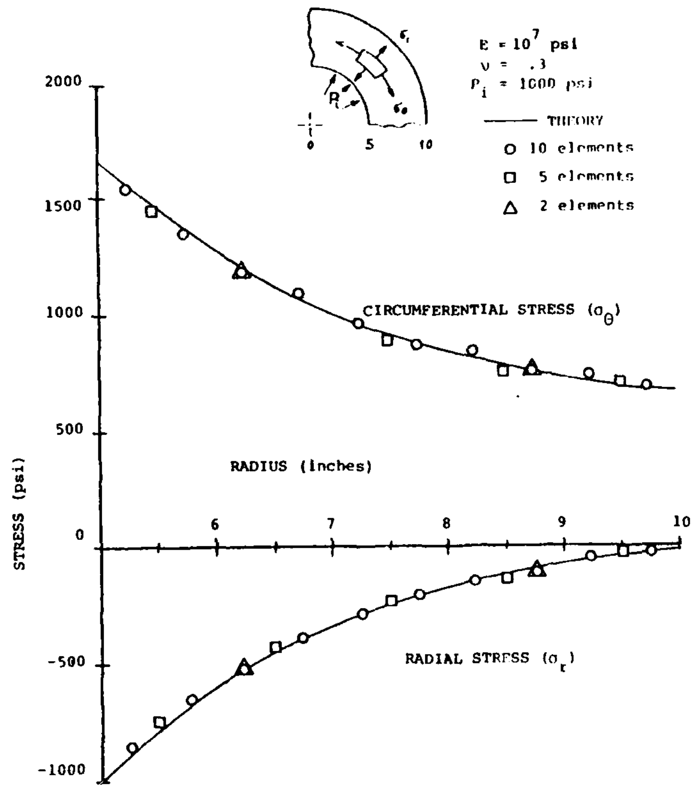



FIGURE 7.6-2. STRESSES IN THICK-WALLED CYLINDER - TRAPRG VS. THEORY.

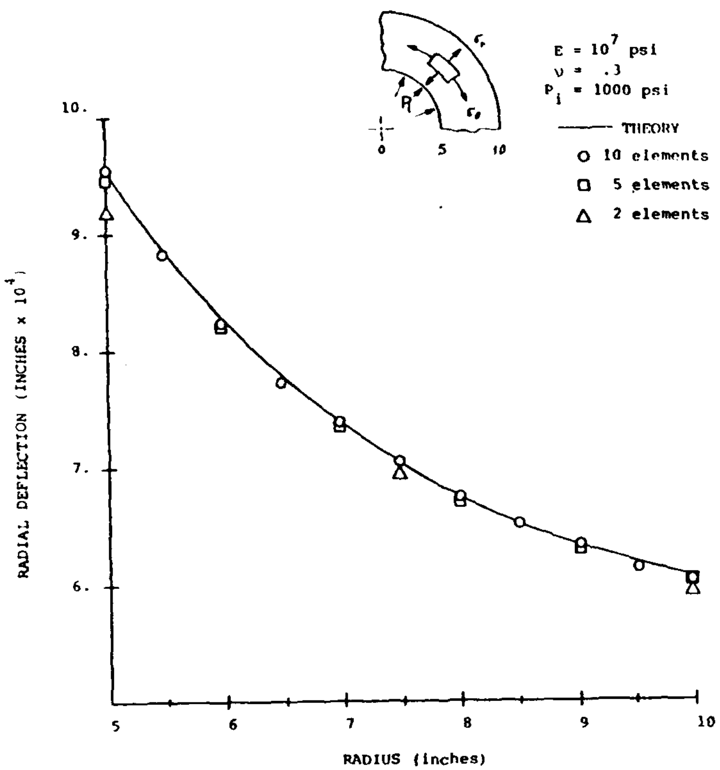



FIGURE 7.6-3. RADIAL DEFLECTION OF THICK-WALLED CYLINDER - TRAPAX VS. THEORY.

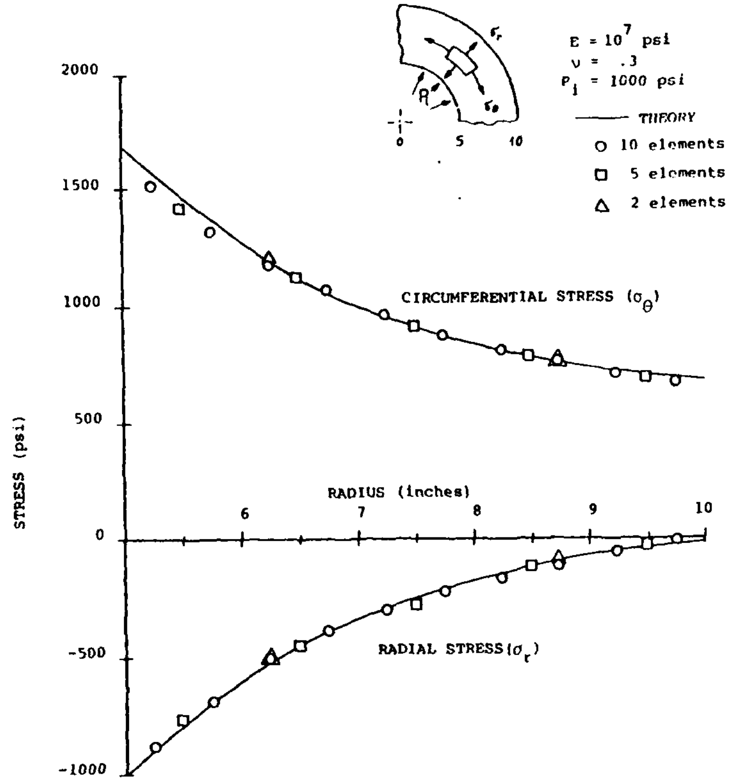



FIGURE 7.6-4. STRESSES IN THICK-WALLED CYLINDER - TRAPAX VS. THEORY.

7.7. The HEXA1, HEXA2, IHEX1, and IHEX2 Solid Elements

COSMIC/NASTRAN has four constant strain solid elements of various shapes (TETRA, WEDGE, HEXA1, HEXA2) and three isoparametric solid elements of 8, 20, and 32 nodes (IHEX1, IHEX2, IHEX3). We decided to evaluate only four of these, and compare their accuracy and performance together in two test problems: a slender cantilevered beam under four different loading conditions; and a pressurized thick-walled cylinder. In addition to these two static test problems, we also studied the eigenvalues of a one-inch cube, using the available eigenvalue extraction schemes.

7.7.1. <u>Theory</u>

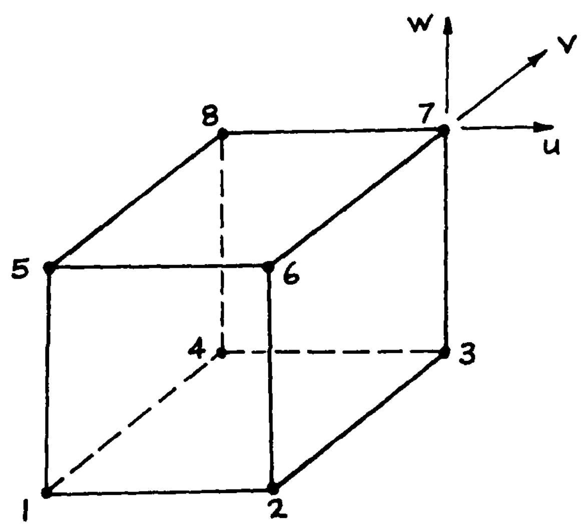
7.7.1.1. <u>HEXA1 and HEXA2 Constant Strain Hexahedral Elements</u>

The HEXA1 and HEXA2 solid elements are comprised of five subtetrahedra and ten overlapping subtetrahedra respectively, each of which is merely a three-dimensional equivalent of the constant strain triangle. Each of the eight nodes in the hexahedron thus consists of three translational degrees of freedom, for a total of 24 DOF's. These elements are subject to the following restrictions: constant strain in each subtetrahedron; uniform and isotropic material properties; uniform temperature in each tetrahedral subelement; translational degrees of freedom only at each node; and no capability for differential stiffness, buckling, and piecewise linear analyses. The assumed displacement field at each node is therefore linear in each Cartesian coordinate:

$$u = q_1 + q_2 + q_3 y + q_4 z$$

$$v = q_5 + q_6 x + q_7 y + q_8 z$$

$$w = q_9 + q_{10} x + q_{11} y + q_{12} z$$



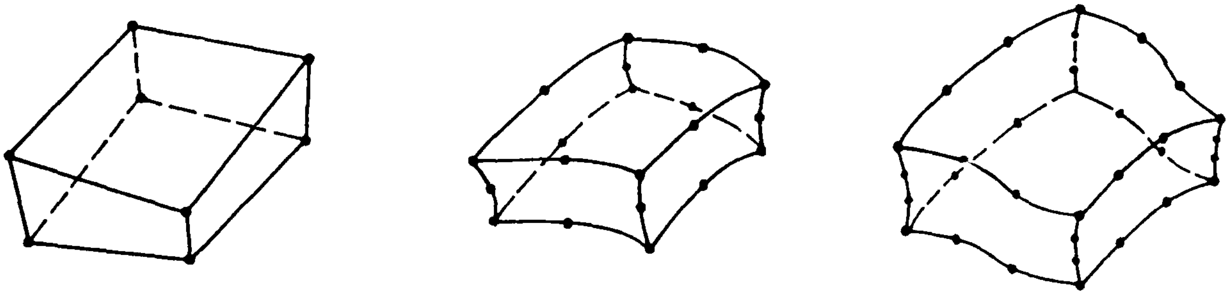
This assumed displacement field produces a constant (uniform) strain and stress in each element.

A hexahedron can be cut into five subtetrahedra into only two different ways. HEXA1 represents a single subdivision into five subtetrahedra, while HEXA2 uses the average of the results of the two types of subdivisions. HEXA2 results in symmetrical deformations when symmetrical loads are applied to a symmetrical hexahedron, while HEXA1 does not (Ref. 7). This subtle, but important, distinction renders HEXA1 to be definitely inferior to HEXA2 and the IHEXi elements, especially in problems where the out-of-plane deflections (or mode shapes) are important. The spurious deflections are discussed further in Subsection 7.7.2.1.

In actual practice, numerous researchers have all concluded that far too many constant strain solid elements are required in a three-dimensional analysis to obtain acceptable results (see for instance, References 48 to 50). From a practical standpoint, these constant strain solid elements are obsolete by current 1980 standards, should be omitted in a modern general-purpose finite element code, and have been rendered inferior by the isoparametric solid elements since the late sixties and the early seventies.

7.7.1.2. <u>IHEX1 and IHEX2 Isoparametric Hexahedral Elements</u>

The IHEXi isoparametric hexahedral solid elements are effective for analyzing a three-dimensional continuum. The material must have isotropic temperature-dependent material properties. "Isoparametric" means the element deformations are represented with the identical interpolating functions used to define the geometry. COSMIC/NASTRAN offers the user a choice of three isoparametric hexahedron solid elements: 8-node (IHEX1), 20-node (IHEX2) and 32-node (IHEX3). These correspond to assumed displacement variations of linear, quadratic, and cubic shapes.

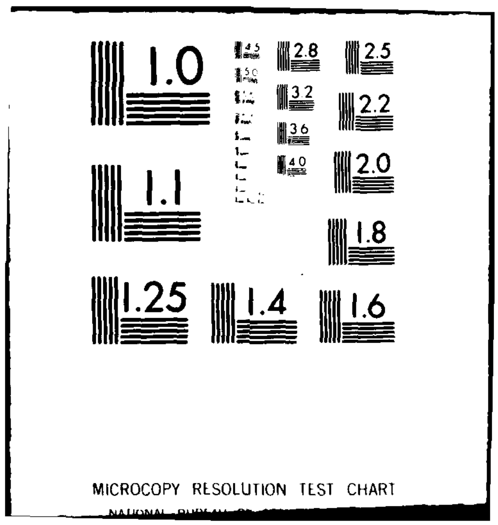


LINEAR (8 nodes) QUADRATIC OR PARABOLIC (20 nodes) CUBIC (32 nodes)

AD-A092 210 SWANSON SERVICE CORP HUNTINGTON BEACH CA F/G 13/13
EVALUATION OF THE NASTRAN GENERAL PURPOSE COMPUTER PROGRAM.(U)
AUG 80 J W JONES, H H FONG, D A BLEHM N00014-79-C-0576
UNCLASSIFIED SSC-81980 NL

2 OF 2
AD A
092210

END
DATE
FILMED
1-81
DTIC


1.0
1.1
1.25
1.4
1.6
1.8
2.0
2.2
2.5
2.8
3.2
3.6
4.0
MICROCOPY RESOLUTION TEST CHART

Clough (Ref. 48) and subsequent researchers have evaluated various three-dimensional solid elements, concluding that the isoparametric elements are superior to other solid elements. The 8-node and 20-node isoparametric solid elements now appear in all general purpose finite element codes. Experience has shown that the 8-node linear element is recommended for problems with high shear deformations and stresses, while the 20-node quadratic element appears to work best in problems with plate or beam bending type deformations. Of these three elements, the 20-node quadratic isoparametric solid element apparently offers the best combination of accuracy, efficiency, and cost, and may well be the most often used solid element today.

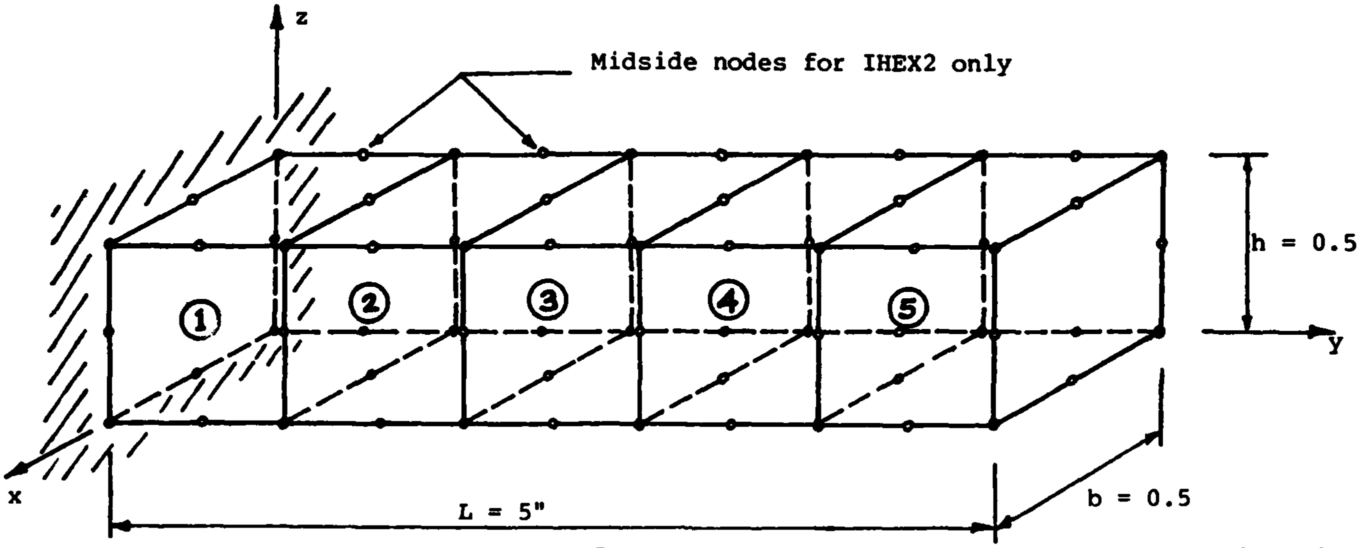
The integrals in the isoparametric element stiffness, mass, and load matrices are evaluated by the use of numerical integration. Gaussian quadrature is used in NASTRAN, and the user has a choice of a 2 x 2 x 2 Gaussian integration scheme or 3 x 3 x 3 integration scheme. For most problems, the 2 x 2 x 2 scheme seems to work best for the linear element (IHEX1), and a 3 x 3 x 3 scheme works best for the two higher order elements. In our advanced evaluation, we evaluated only HEXA1, HEXA2, IHEX1, and IHEX2.

7.7.2. <u>Advanced Evaluation</u>

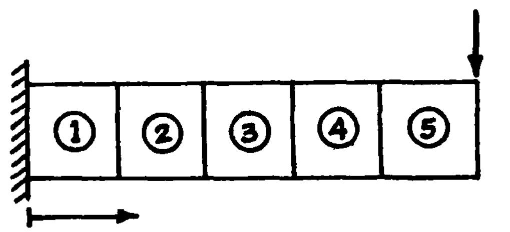
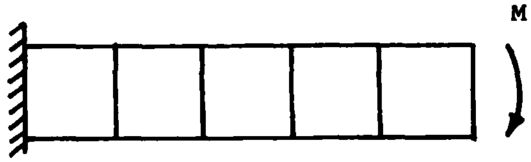
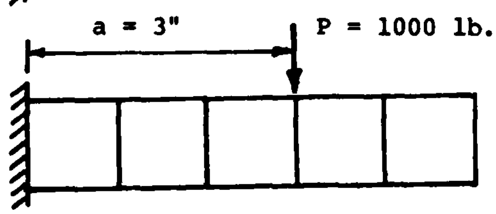
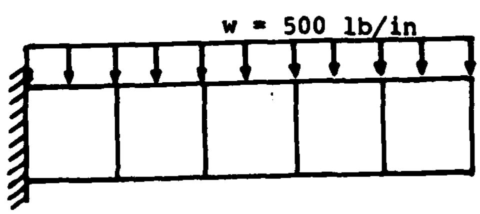
The two test cases used to evaluate HEXA1, HEXA2, IHEX1, and IHEX2 were a slender cantilevered beam (with four loading conditions) and an internally pressurized thick-walled cylinder. The cantilevered beam problem is commonly used to test the ability of a solid element to represent bending behavior. The thick-walled cylinder problem enables us to assess the behavior of the solid elements in an enforced radial dilatation mode. A third test problem, eigenvalue extraction of a unit cube, allows us to verify both NASTRAN solid element theory and eigenvalue extraction efficiency.

7.7.2.1. Cantilevered Beam

The four loading cases for the cantilevered beam are:




$E = 10^7$ psi, $L = 5$ in., $I = \frac{1}{12}(.5)^4 = 5.2083 \times 10^{-3}$ in.$^4$

<table>
<tr><th>CASE</th><th>LOADING</th><th>DEFLECTION $\delta_z$</th></tr>
<tr><td>1</td><td>
</td><td>$\frac{Px^2}{6EI}(3L-x)$</td></tr>
<tr><td>2</td><td>
</td><td>$\frac{Mx^2}{2EI}$</td></tr>
<tr><td>3</td><td>
</td><td>$\frac{Px^2}{6EI}(3a-x) \quad 0<x<a$<br>$\frac{Pa^2}{6EI}(3x-a) \quad a<x<L$</td></tr>
<tr><td>4</td><td>
</td><td>$\frac{wx^2}{24EI}(x^2+6L^2-4Lx)$</td></tr>
</table>

Theoretical and finite element deflections and stresses are plotted in Figures 7.7-1 to 7.7-4. It is obvious from these plots that of the four solid elements evaluated in this bending example, the only element which performed reasonably well is IHEX2. IHEX2 consistently produced good correlation with theoretical deflections and stresses in all four load cases.

The next best solid element is IHEX1, but it gave deflections and stresses only 30 to 50 percent of those predicted by theory. HEXA1 and HEXA2 both correlated very poorly with theoretical deflections. This lackluster performance of IHEX1, HEXA1, and HEXA2 in a beam bending test problem can probably be attributed to the fact we used only one layer of elements through the thickness. This leads us to the conclusion that, for these three solid elements, the analyst should use more than one element through the thickness in a beam bending problem before he can hope for reasonably accurate answers. However, we note that for a similar problem, MSC/NASTRAN's 6- and 16-node PENTA and 8- and 20-node HEXA isoparametric solid elements as well as STARDYNE's 8-node isoparametric solid element all correlated fairly well with theoretical displacements even for a beam model with one layer through the thickness.

Our results for HEXA1 in the cantilevered beam benchmark also indicated the existence of spurious modes in the lateral x-direction when the primary deflection occurs in the z-direction. These lateral sidesway modes for HEXA1 were attributed to its composition of five subtetrahedra and the basic unsymmetrical nature of this composition. The magnitude of this sidesway deflection was approximately five to ten percent of the z-deflection.

The effects of modeling a cantilevered beam with more than one layer in depth and in width, and slender versus short deep beams have already been discussed elsewhere by several researchers (e.g. References 48,50,54). These references cite the superiority of the 20-node isoparametric solid element in slender beam bending problems over 8-node isoparametric solid elements and constant strain as well as higher order hexahedra. However, for short deep cantilever beams where shear deformation is important, the 8-node isoparametric solid is better than the 20- node isoparametric solid and constant strain or higher-order hexahedra.

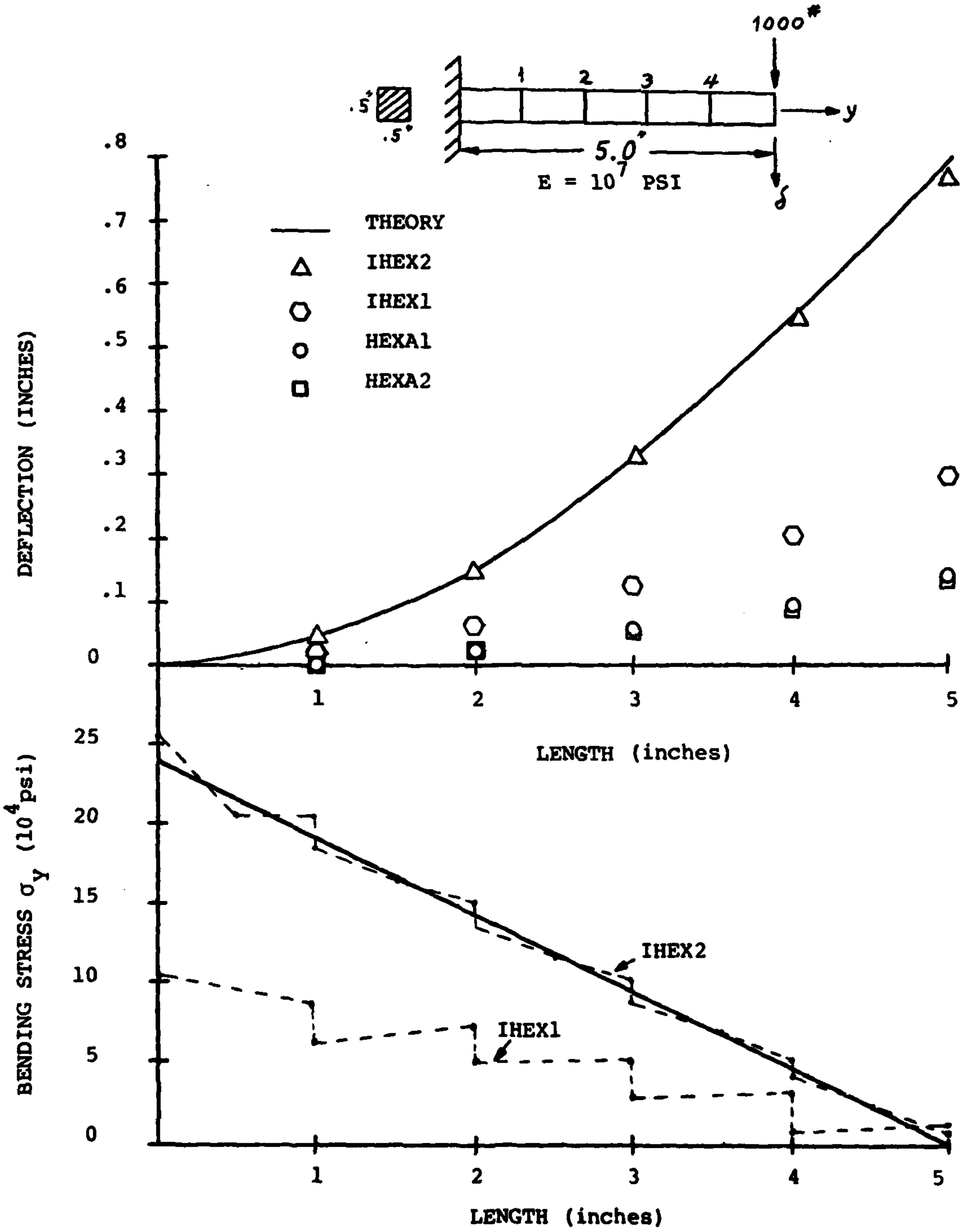



FIGURE 7.7-1. RESULTS OF CANTILEVERED BEAM TEST OF SOLID ELEMENTS - CASE 1. END LOAD.

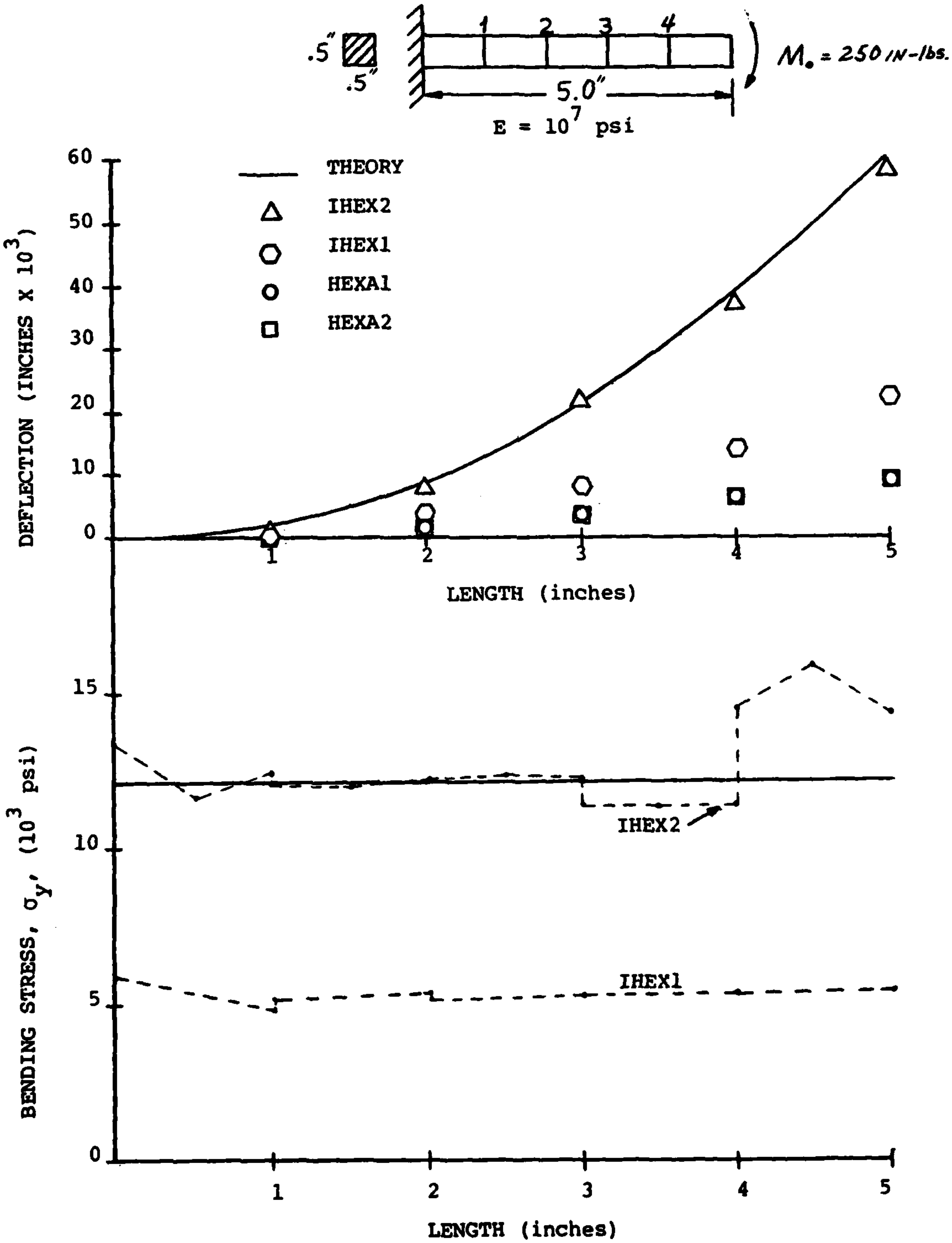



FIGURE 7.7-2. RESULTS OF CANTILEVERED BEAM TEST FOR SOLID ELEMENTS - CASE 2. END MOMENT.

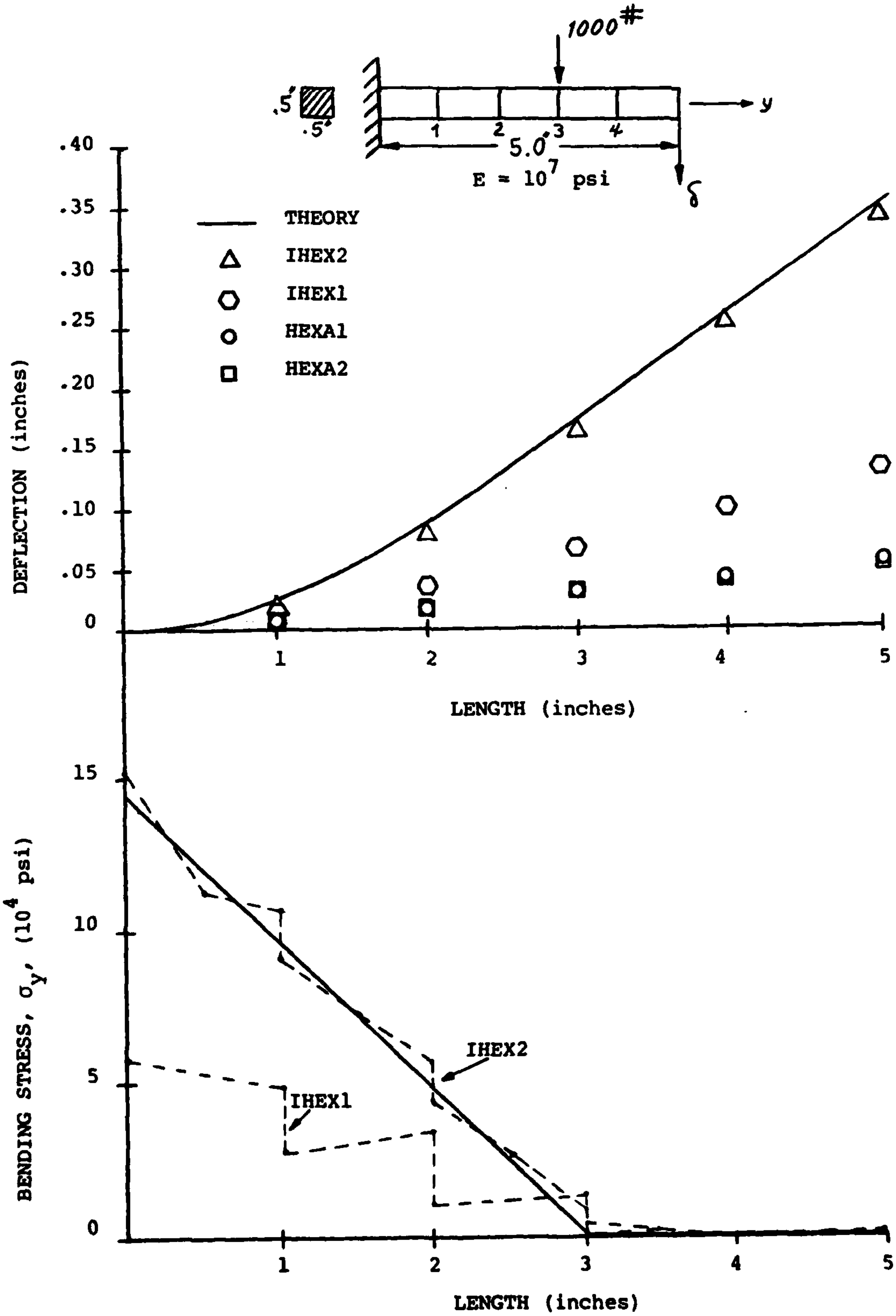



FIGURE 7.7-3. RESULTS OF CANTILEVERED BEAM TEST FOR SOLID ELEMENTS - CASE 3. INTERMEDIATE LOAD.

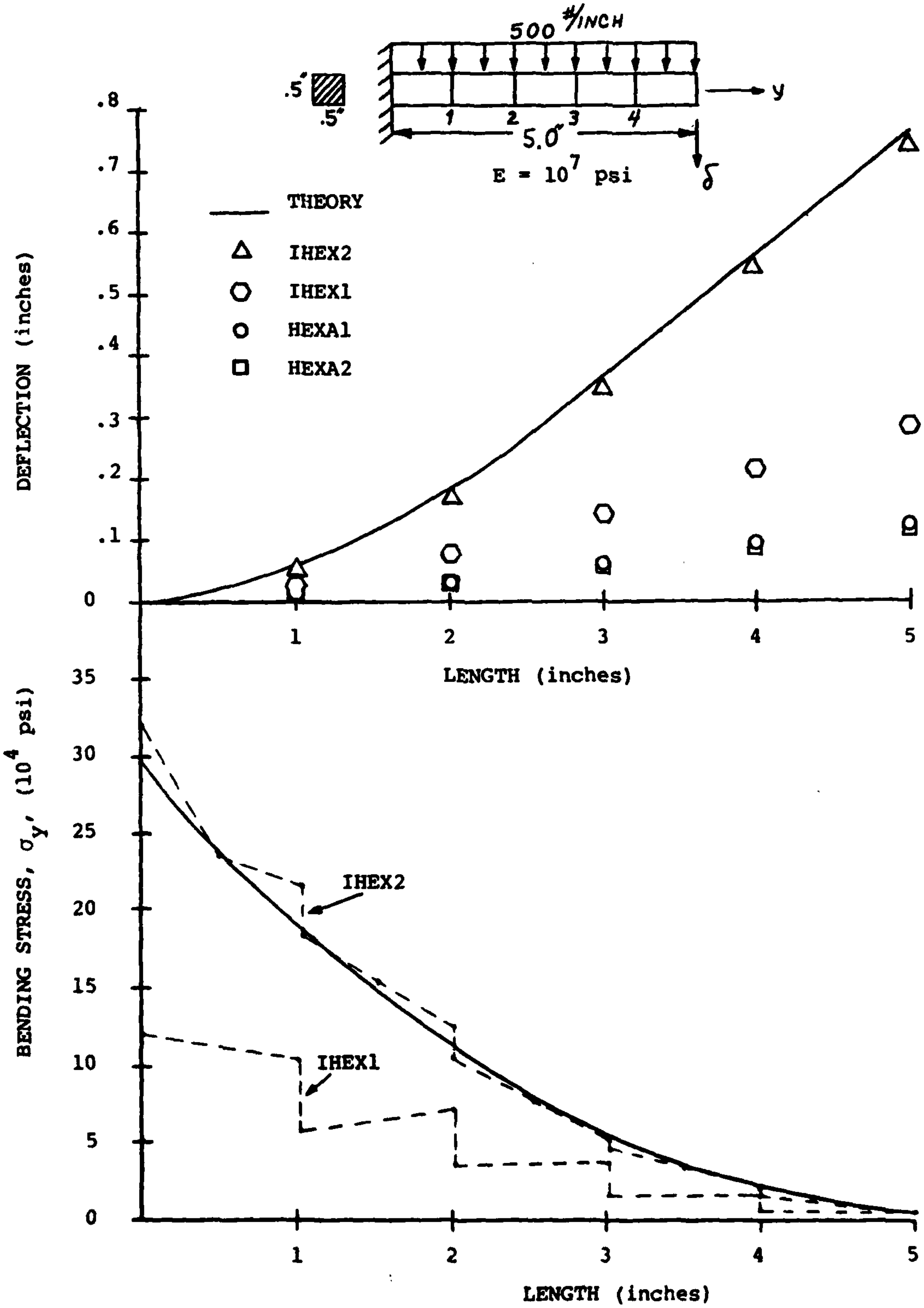



FIGURE 7.7-4. RESULTS OF CANTILEVERED BEAM TEST FOR SOLID ELEMENTS - CASE 4. UNIFORM LOAD.

NASTRAN does not offer the analyst an option to improve solution accuracy by the use of "incompatible bending modes" (Ref. 55). These so-called "bubble modes" have often been used effectively in bending problems; they improve microscopic equilibrium within the element, though violating interelement compatibility. The use of such a device has been used very successfully to improve the general performance of the 8-node isoparametric solid element (for instance, the ANSYS STIF 45 element).

For the NASTRAN user, therefore, the only solid element which should be used in bending problems is IHEX2, the 20-node isoparametric solid element. Use of a 20-node element implies a more complex and time-consuming input, leading to a higher probability of input error. IHEX3 (the 32-node isoparametric solid element) was not evaluated in this study but the same comments apply.

7.7.2.2. Thick-Walled Cylinder

This test problem is identical to that used for TRAPRG (Subsection 7.6). The problem measures the ability of the solid elements to represent a dilatational type of deformation (in the radial direction only). The displacement results for the four solid elements are shown in Figures 7.7-5 to 7.7-8, and the radial and hoop stress correlations with theory are shown in Figures 7.7-9 to 7.7-12. In this problem, three mesh sizes were used: 2, 5, and 10 elements through the cylinder thickness. This time, the radial displacement correlations are usually 12 percent or better, and the stress correlations with theory were all remarkably good. Even though the thick-walled cylinder test problem is not as severe a test on the solid elements as a cantilevered beam, the outstanding performance of all four NASTRAN solid elements is still remarkable, even for the 2-element model.

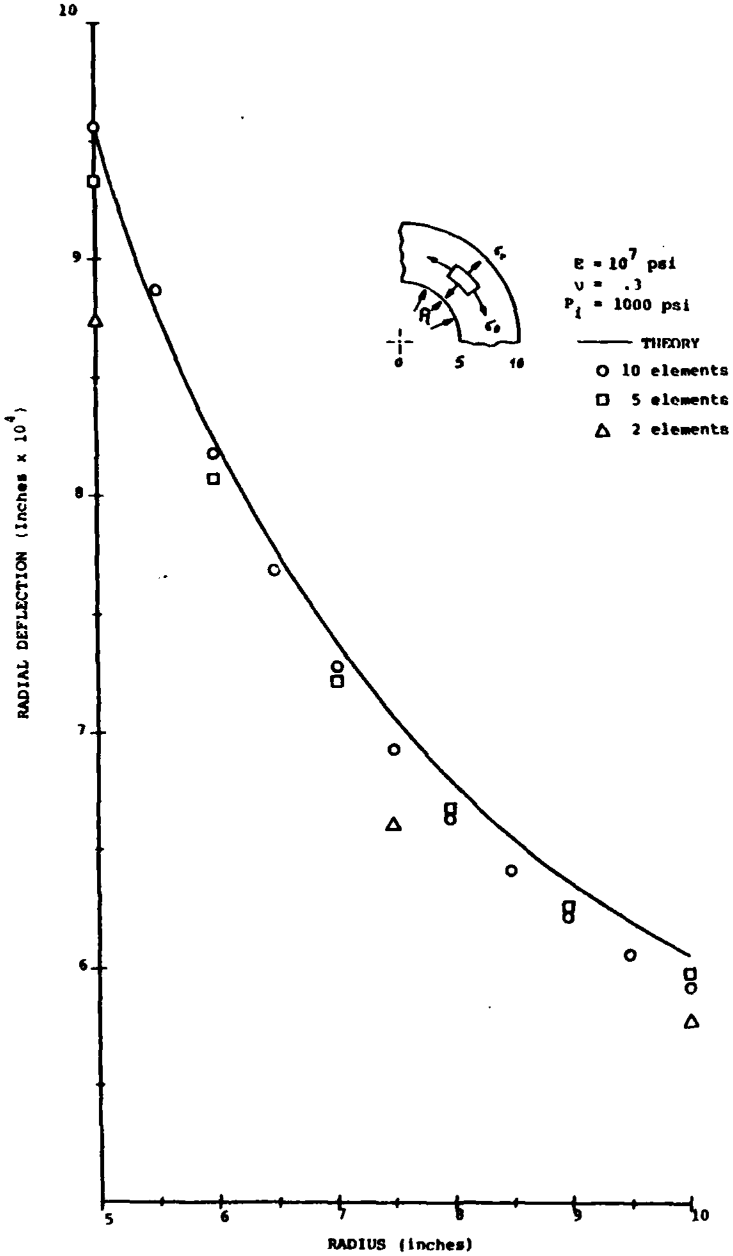



FIGURE 7.7-5. HEXA1 THICK-WALLED CYLINDER RESULTS - DEFLECTIONS.

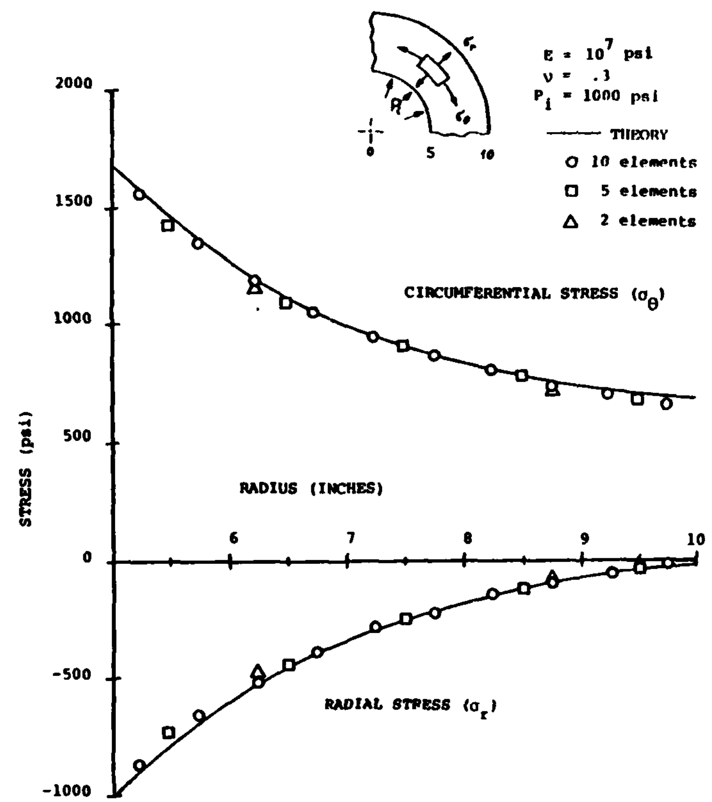



FIGURE 7.7-6. HEXA1 THICK-WALLED CYLINDER RESULTS-STRESSES.

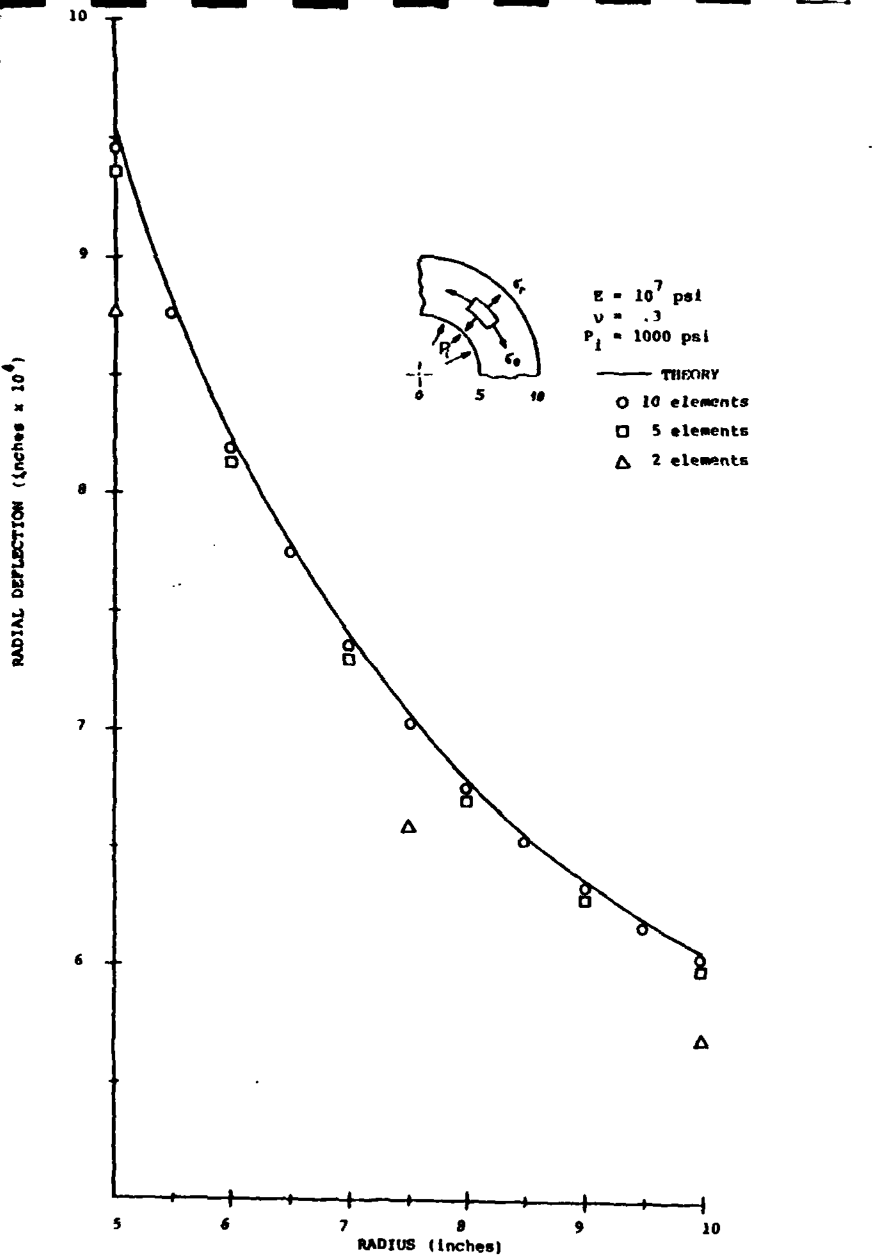



FIGURE 7.7-7. HEXA2 THICK-WALLED CYLINDER RESULTS DEFLECTIONS.

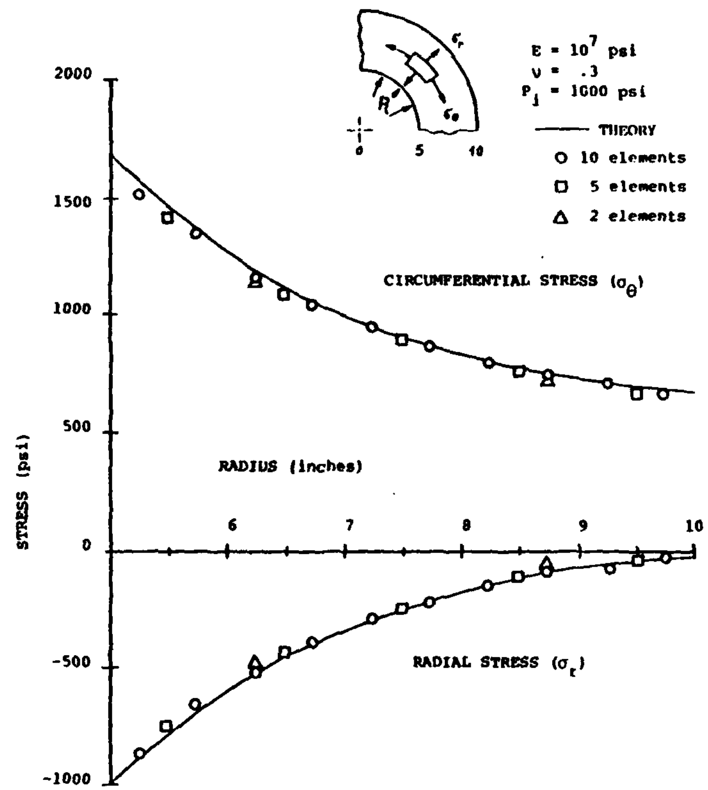



FIGURE 7.7-8. HEXA2 THICK-WALLED CYLINDER RESULTS - STRESSES.

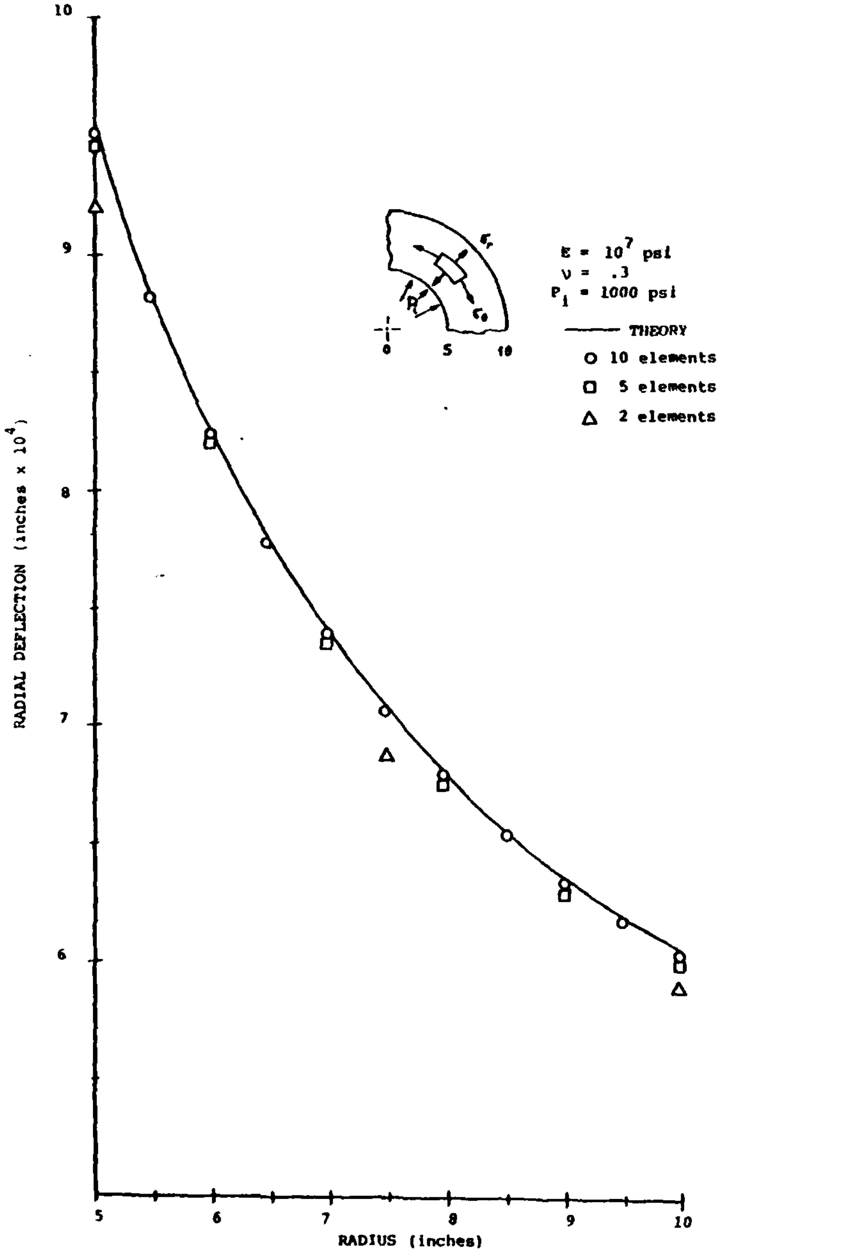



FIGURE 7.7-9. IHEX1 THICK-WALLED CYLINDER RESULTS - DEFLECTIONS.

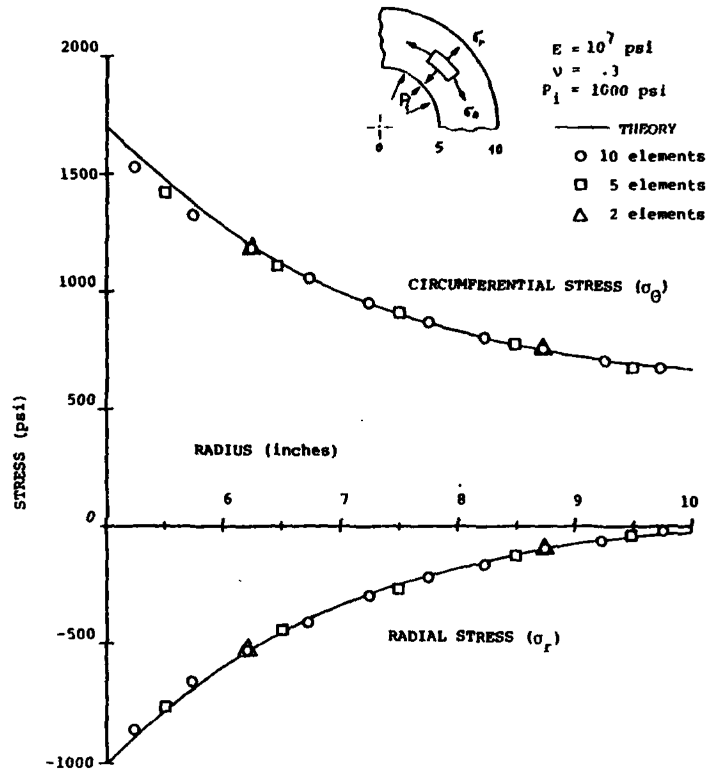



FIGURE 7.7-10. IHEX1 THICK-WALLED CYLINDER RESULTS - STRESSES.

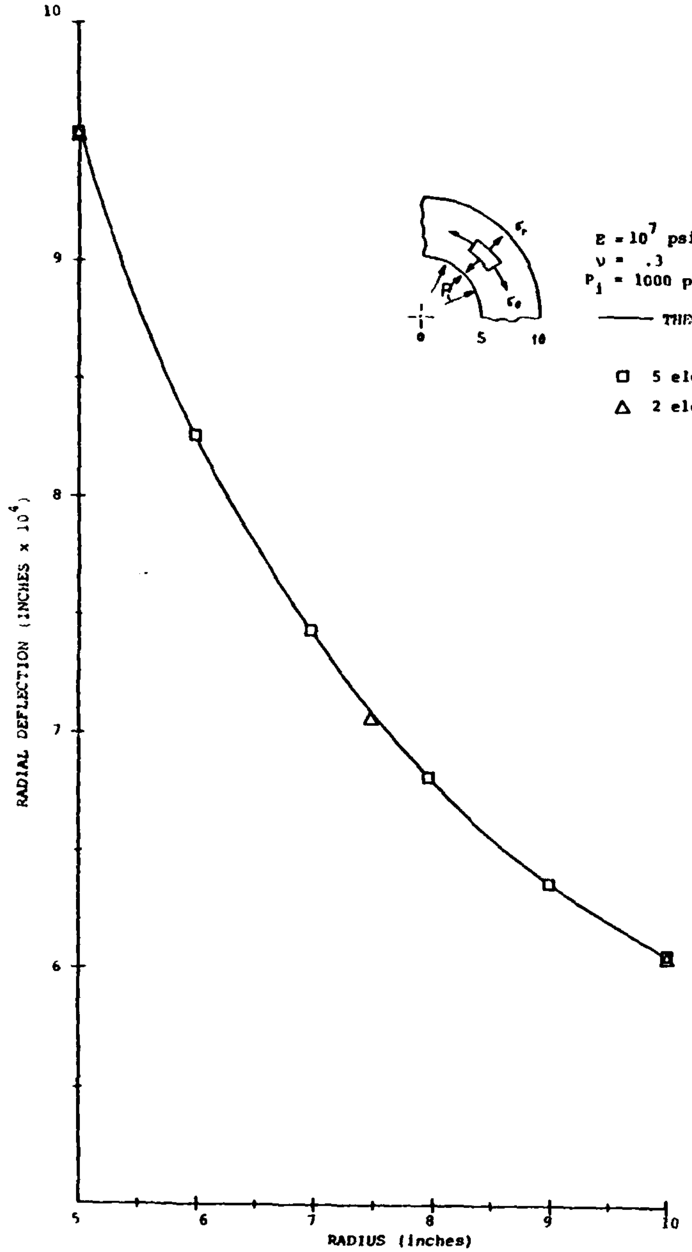



FIGURE 7.7-11. IHEX2 THICK-WALLED CYLINDER RESULTS - DEFLECTIONS.

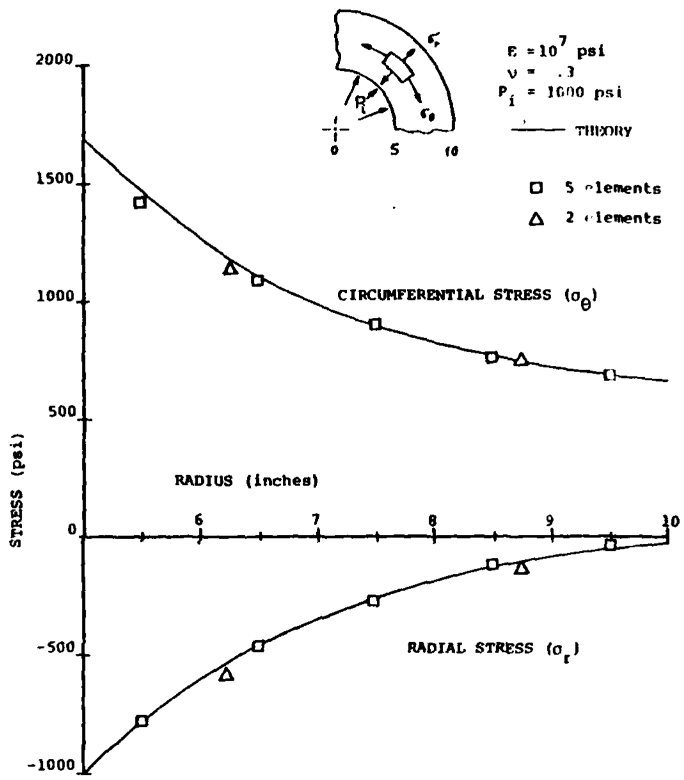



FIGURE 7.7-12. IHEX2 THICK-WALLED CYLINDER RESULTS - STRESSES.

7.7.2.3. Eigenvalue Extraction

The NASTRAN solid elements HEXA1, HEXA2 and IHEX1 were subjected to three different eigenvalue extraction methods in a third type of element test. The use of an eigenvalue and vector solution has been used to study the convergence characteristics of an element (Ref. 51), though, to our knowledge, this procedure has previously been applied only to two-dimensional elements. It has been shown that, provided that the monotonic convergence conditions are satisfied, the finite element analysis overestimates the stiffness of the system, i.e., the calculated displacements are smaller (in some norm) than the exact displacements. Hence when studying the effectiveness of an element, we need to investigate its stiffness characteristics, and an element that is less stiff will be more effective. One way of investigating the element stiffness characteristics is to represent the element stiffness matrix on the basis of its eigenvector, namely, solving the eigenvector problem

$$( [K] - \lambda I)\ \phi = 0 \quad .$$

The results of this test are important in several respects. First, the eigenvalue extraction methods available in NASTRAN may be compared to each other and to a known solution. The eigenvalues are expected to contain rigid body modes ( three translations and three rotations) and several repeated roots, thus posing a good test of the eigenvalue extraction methods for both accuracy and efficiency. Secondly, the problem was solved as if it were a modal analysis. Two different approaches were used in this regard. The first method was to require the program to lump the mass according to the particular element algorithm. The second was to give the element a zero density and place lumped masses at each node. Finally, the mode shapes produced from the eigenvalue extraction should be distinct and recognizable, i.e., shear, flexure, dilatational, etc. Failure to produce distinct mode shapes indicates that spurious displacement modes exist in the element formulation.

The eigenvalues obtained for these elements are shown in Tables 7.7-1 through 7.7-3. The columns labeled "unit cube" give the results for one solid element, one inch on each side. (A density of 1.0 slugs, Young's modulus of 10 x $10^6$ psi and Poisson's ratio of .3 were used for material properties). The first

| EXTRACTION METHOD | | UNIT CUBE | | UNEQUAL CUBE | UNEQUAL & LUMPED MASS |
|---|---|---|---|---|---|
| | | GIVENS | INVERSE | GIVENS | GIVENS |
| Mode | 7 | 707 | 707 | 707 | 596 |
| Number | 8 | 707 | 707 | 707 | 599 |
| | 9 | 821 | 821 | 818 | 741 |
| | 10 | 821 | 821 | 821 | 744 |
| | 11 | 821 | 821 | 825 | 744 |
| | 12 | 1102 | 1102 | 1100 | 882 |
| | 13 | 1102 | 1102 | 1104 | 882 |
| | 14 | 1148 | 1148 | 1147 | 882 |
| | 15 | 1148 | 1148 | 1148 | 939 |
| | 16 | 1148 | 1148 | 1149 | 942 |
| | 17 | 1184 | 1184 | 1185 | 944 |
| | 18 | 1184 | 1184 | 1185 | 964 |
| | 19 | 1184 | 1184 | 1185 | 973 |
| | 20 | 1323 | 1323 | 1323 | 1139 |
| | 21 | 2043 | 2043 | 2043 | 1606 |
| | 22 | 2152 | 2152 | 2151 | 1607 |
| | 23 | 2152 | 2152 | 2152 | 1609 |
| | 24 | 2152 | 2152 | 2153 | 1762 |

TABLE 7.7-1. EIGENVALUES FOR HEXA1 ELEMENT (HERTZ)

| EXTRACTION METHOD | | UNIT CUBE | | | | UNEQUAL CUBE | | UNEQUAL & LUMPED MASS | |
|---|---|---|---|---|---|---|---|---|---|
| | | GIVENS | * | INVERSE | * | GIVENS | * | GIVENS | * |
| Mode | 7 | 721 | + | 721 | + | 719 | T | 719 | T |
| Num-ber | 8 | 721 | + | 721 | + | 723 | T | 721 | T |
| | 9 | 882 | + | 882 | + | 878 | E | 878 | E |
| | 10 | 882 | + | 882 | + | 878 | SH | 878 | SH |
| | 11 | 882 | + | 882 | + | 883 | FX1 | 883 | FX1 |
| | 12 | 882 | + | 882 | + | 883 | FX1 | 883 | FX1 |
| | 13 | 882 | + | 882 | + | 883 | FX1 | 883 | FX1 |
| | 14 | 882 | + | 882 | + | 883 | SH | 883 | SH |
| | 15 | 882 | + | 882 | + | 883 | SH | 883 | SH |
| | 16 | 882 | + | 882 | + | 883 | E | 883 | E |
| | 17 | 1195 | + | 1195 | + | 1190 | H | 1190 | H |
| | 18 | 1195 | + | 1195 | + | 1195 | H | 1195 | H |
| | 19 | 1195 | + | 1195 | + | 1200 | H | 1200 | H |
| | 20 | 1349 | + | 1349 | + | 1345 | FX2 | 1345 | FX2 |
| | 21 | 1349 | + | 1349 | + | 1349 | FX2 | 1349 | FX2 |
| | 22 | 1349 | + | 1349 | + | 1352 | FX2 | 1352 | FX2 |
| | 23 | 1372 | + | 1372 | + | 1372 | FX3 | 1372 | FX3 |
| | 24 | 1591 | B | 1591 | + | 1591 | B | 1591 | B |

*Produced mode shape of type indicated by letter (see Figure 7.7-13).

+Mode Shape Nondistinct.

TABLE 7.7-2. EIGENVALUES FOR HEXA2 ELEMENT (HERTZ).

| EXTRACTION METHOD | | UNIT CUBE GIVENS | * | UNIT CUBE INVERSE | * | UNEQUAL CUBE GIVENS | * | UNEQUAL & LUMPED MASS OF NODES GIVENS | * |
|---|---|---|---|---|---|---|---|---|---|
| Mode | 7 | 856 | + | 856 | + | 849 | | 356 | T |
| Number | 8 | 856 | + | 856 | + | 863 | | 364 | T |
| | 9 | 1255 | + | 1255 | + | 1255 | | 486 | H |
| | 10 | 1448 | + | 1448 | + | 1448 | | 489 | H |
| | 11 | 1448 | + | 1448 | + | 1449 | | 489 | H |
| | 12 | 1529 | + | 1529 | + | 1528 | | 622 | FX1 |
| | 13 | 1529 | + | 1529 | + | 1529 | | 622 | FX1 |
| | 14 | 1647 | + | 1647 | + | 1644 | | 622 | FX1 |
| | 15 | 1647 | + | 1647 | + | 1651 | | 718 | FX3 |
| | 16 | 1670 | + | 1670 | + | 1671 | | 876 | E |
| | 17 | 1873 | + | 1873 | + | 1873 | | 876 | SH |
| | 18 | 2163 | + | 2163 | + | 2163 | | 879 | FX2 |
| | 19 | 2163 | + | 2163 | + | 2163 | | 882 | SH |
| | 20 | 2239 | + | 2239 | + | 2239 | | 882 | FX2 |
| | 21 | 2634 | + | 2634 | + | 2632 | | 885 | FX2 |
| | 22 | 2649 | + | 2649 | + | 2648 | | 885 | SH |
| | 23 | 2649 | + | 2649 | + | 2654 | | 885 | E |
| | 24 | 2920 | + | 2920 | + | 2921 | | 1591 | B |

*Produced mode shape of type indicated by letter (see Figure 7.7-13).

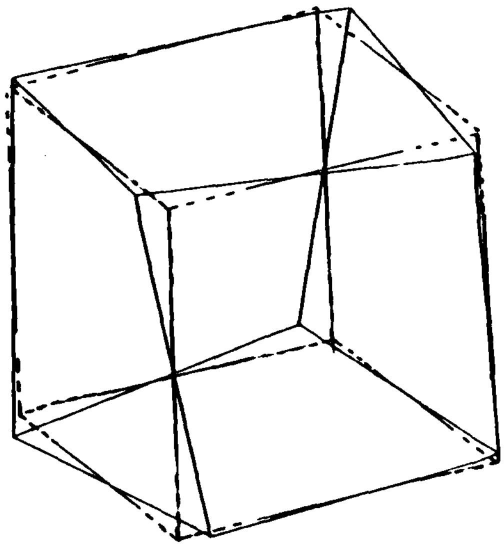
+Mode shape nondistinct.

TABLE 7.7-3. EIGENVALUES FOR IHEX1 ELEMENT (HERTZ).

six eigenvectors were calculated in all cases to be essentially zero frequency (rigid body modes) and therefore are not reported. As many as eight repeated roots were found in some cases. It is interesting to note that the Givens method was by far the most efficient method for this problem. The typical cost of one run was approximately $15 using the Givens method. The Inverse method, by comparison, generally cost approximately $45 per run. The Determinant method of solution was attempted for all three NASTRAN elements. In all cases the time limit for the run was exceeded (a cost of approximately $150) and no solution was obtained. The results obtained by the Givens and Inverse method were essentially identical, therefore, the Givens method was used for the majority of the runs made.

The eigenvectors produced by using the exact cube dimensions were not well-defined, i.e., the mode shapes did not correspond to intuitively expected forms, such as extension, shear, etc. This is probably because of the non-uniqueness of the eigenvectors formed from the unit cube geometry. In order to accentuate the shapes, an "unequal cube" geometry was run. The x dimension was made 0.99 inch, y was made 1.01 inches while z was kept at exactly one. This model produced eigenvalues which are essentially equal to those produced by the unit cube but, in the case of HEXA2, produced well-defined eigenvectors. The shapes are shown in Figures 7.7-13(a) and 7.7-13(b). The mode shapes can be described as follows:

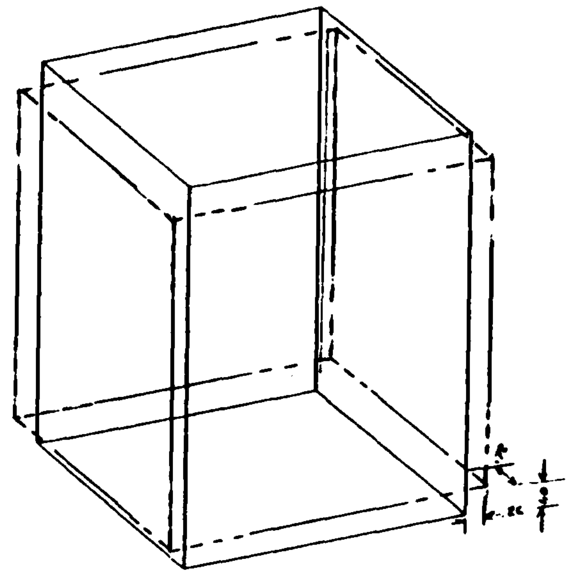
| | |
|---|---|
| MODE I (Type T) | A twist-type mode where one face of the cube is rotated with respect to the opposite face. This mode appears to be the lowest for all elements. There are two repeated eigenvectors with this mode shape. This mode is designated by a "T" in the Tables. |
| MODE II (Type E) | An extension mode in which the cube elongates in one direction and is reduced in the other two directions. The volume is preserved. There are two repeated eigenvectors and two distinct mode shapes of this type. This mode is designated by an "E" in the Tables. |
| MODE III (Type FX1) | This appears to be a flexure mode in which two opposite sides of the cube are transformed into rectangles each having the longer sides perpendicular to the other. Three modes of this type were observed. This mode is designated by a FX1 in the Tables. |



MODE I

TYPE T

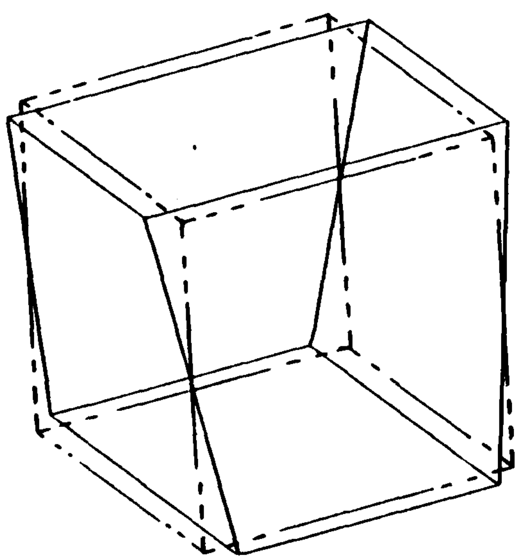
2 REPEATED ROOTS



MODE II

TYPE E

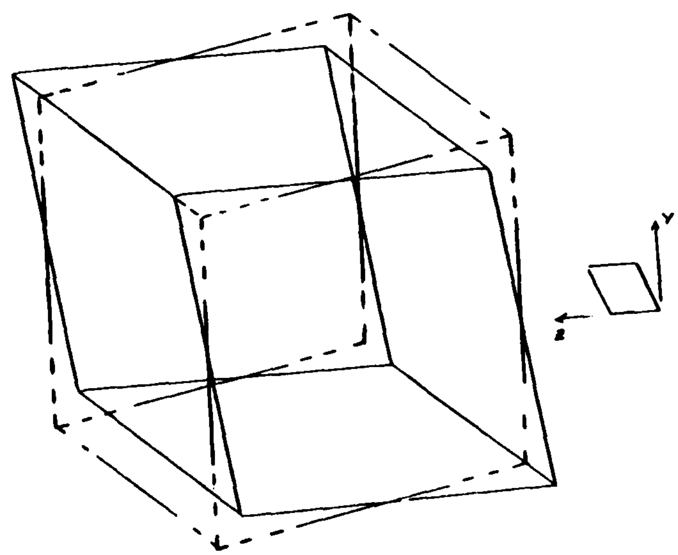
2 REPEATED ROOTS



MODE III

TYPE FX1

3 REPEATED ROOTS

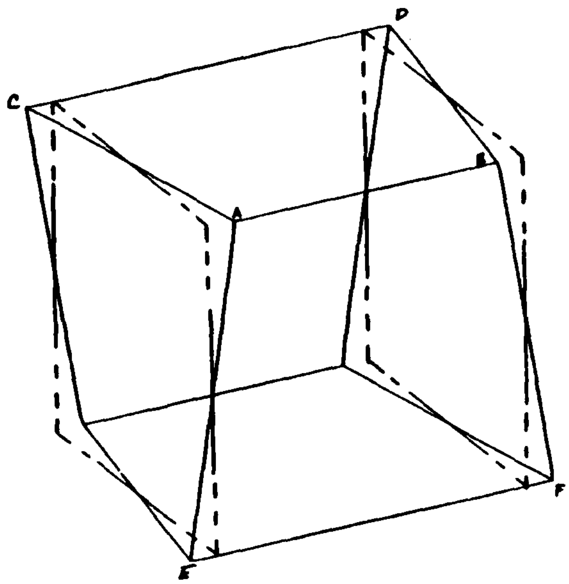


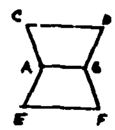

MODE IV

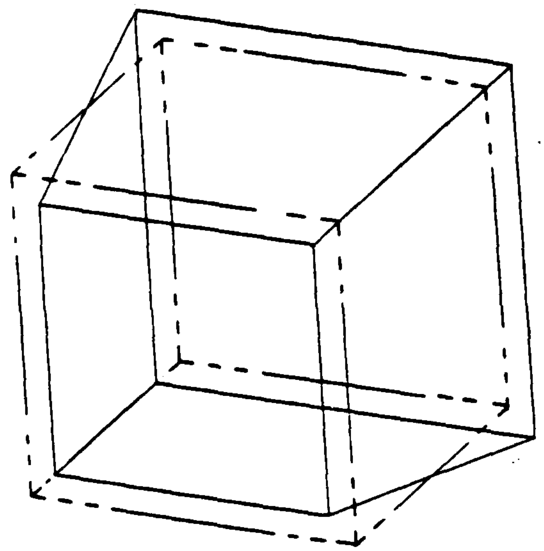
TYPE SH

3 REPEATED ROOTS

FIGURE 7.7-13(a). EIGENVECTORS FOR 8-NODE HEXAHEDRON.



MODE V

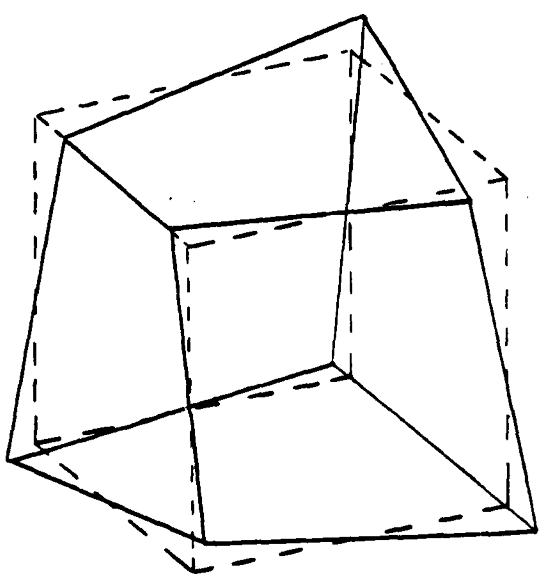
TYPE H

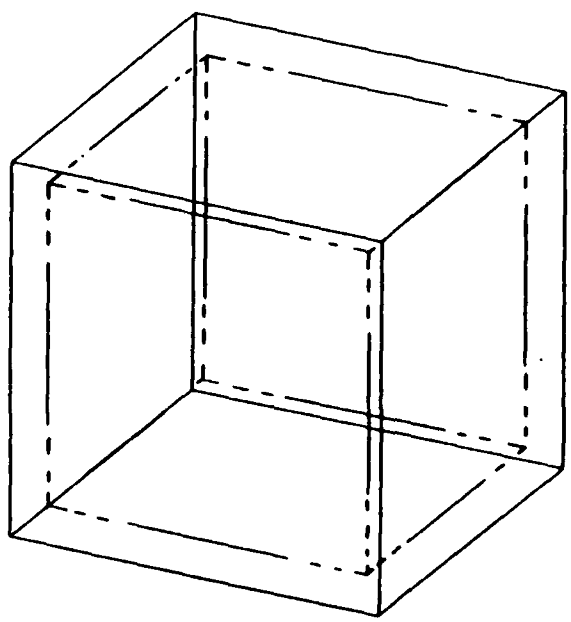
3 REPEATED ROOTS

MODE VI

TYPE FX2

3 REPEATED ROOTS





MODE VII

TYPE FX3

1 ROOT

MODE VIII

TYPE B

1 ROOT

FIGURE 7.7-13(b). EIGENVECTORS FOR 8-NODE HEXAHEDRON.

| | |
|---|---|
| MODE IV (Type SH) | This is a shear mode in which opposite sides remain parallel but a shearing motion is exhibited in each of three planes. Three such modes were found. The designation used in the Tables is SH. |
| MODE V (Type H) | This mode was designated an hourglass mode (H is the symbol used in the Tables). The shape is characterized by non-zero deflections only in one direction, i.e., ux motion only, etc. Opposite corners of the cube have identical motion. When viewed looking at one edge, the deformed shape looks somewhat like an hourglass. Three such modes were found. |
| MODE VI (Type FX2) | A second flexural mode was identified in which the cube dilated on one face and contracted on an opposite face. This was designated as a French Roof mode with the Table designation FX2. Three repeated eigenvalues exhibited this mode. |
| MODE VII | This mode is a third flexural mode in which two opposite faces each form a diamond pattern with the long axis of the diamond diagonals mutually perpendicular. Only one eigenvalue/eigenvector exhibited this mode shape. It is termed FX3 for identification in the Tables. |
| MODE VIII (Type B) | The highest mode in all cases appears to be a dilatational mode. This was designated by "B" in the tables (for breathing) and only one such mode was found for each element considered. |

The last column in the Tables gives the results for each element for the case where 1/8 of the element mass is lumped at each node point. Several conclusions can be drawn from these runs.

First, the HEXA1 element did not produce distinct mode shapes for any of the conditions analyzed. It should be noted that this element is composed of 5 tetrahedra and has already been shown to produce spurious deflection shapes (Ref. Section 7.7.1.1). Additionally, the masses are lumped at the nodes on the following basis. The mass calculation involves the calculation of the total mass of each tetrahedron in the element and assigning 1/4 of the total mass to each of the four node points. This method of distributing mass will not produce eight equal masses at the nodes. This procedure results in a dependency on the node numbering scheme, i.e., the results will be different for this element depending upon the order of element node numbering, which is, of course, undesirable. Thus, the element does not produce symmetrically shaped eigenvectors as it should. Further, it is noted that the eigenvalues changed when the masses

were lumped at the nodes by the analyst rather than using the element algorithm. This was not true for HEXA2, which uses the same basic subelement (tetrahedra) and indicates that there may be some error in the programming of the element.

Table 7.7-2 shows the results of these tests for the HEXA2 element. It can be seen that the eigenvalues are almost identical for all cases run. This was expected since the masses are lumped equally at the nodes (because of the 10 overlapping tetrahedra) and the symmetry of the resulting displacement functions within the element. It is noted also that the frequencies are lower than for the HEXA1 element, which indicates that HEXA2 is less stiff and thus should produce better results.

Table 7.7-3 shows the results for IHEX1, the eight-node isoparametric hexahedron. It was found that recognizable mode shapes were not evident until the lumped mass case was run. Even though the documentation stated that a lumped mass formulation is used as a default option in the program, the Programmer's Manual indicates that no provision is made for the computation of a lumped mass matrix for the isoparametric hexahedron elements. Thus, when the eigenvalue problem for this element is solved using the modal analysis rigid format and using a non-zero material density, the mass is calculated according to the shape function for the element. Therefore , in order to compare the element stiffness to the HEXAi elements, the lumped mass cases must be used. On this basis, it is seen that the eigenvalues are lower for IHEX1 than either of the HEXAi elements indicating that the element is less stiff. It is also interesting to note that the mode shapes were not produced in the same order as for the HEXA2 elements.

As a further check on this test, a series of runs were made using a similar element in the ANSYS program, the STIF 45 eight-node isoparametric hexahedron (the incompatible bending modes which are an option in this element were suppressed). The results are shown in Table 7.7-4. As can be seen, the eigenvalues are very close to those found for IHEX1 and the distribution of mode shapes was similar. It is noted that the method of eigenvalue extraction used was the Jacobi method.

| EXTRACTION METHOD | | UNEQUAL CUBE JACOBI | * | UNEQUAL CUBE & LUMPED MASS JACOBI | * |
|---|---|---|---|---|---|
| Mode | 7 | 1068 | T | 356 | T |
| Number | 8 | 1094 | T | 365 | T |
| | 9 | 1520 | E | 486 | H |
| | 10 | 1521 | SH | 488 | H |
| | 11 | 1536 | SH | 490 | H |
| | 12 | 1537 | SH | 507 | FX1 |
| | 13 | 1538 | E | 509 | FX1 |
| | 14 | 1872 | FX1 | 512 | FX1 |
| | 15 | 1873 | FX1 | 592 | FX2 |
| | 16 | 1873 | FX1 | 693 | FX2 |
| | 17 | 2162 | FX3 | 698 | FX2 |
| | 18 | 2536 | H | 721 | FX3 |
| | 19 | 2536 | H | 878 | E |
| | 20 | 2547 | H | 878 | SH |
| | 21 | 2642 | FX2 | 883 | SH |
| | 22 | 2649 | FX2 | 887 | SH |
| | 23 | 2655 | FX2 | 888 | E |
| | 24 | 2757 | B | 1589 | B |

*Produced mode shape of type indicated by letter (see Figure 7.7-13).

TABLE 7.7-4. EIGENVALUES FOR ANSYS STIF 45 ELEMENT (HERTZ).

7.7.3. Conclusions

Our evaluation results indicate that the overall quality, performance, and efficiency of the COSMIC/NASTRAN solid elements are fair--but one must be very careful in selecting the appropriate element and mesh for a particular application.

The performance of each solid element in the evaluation benchmarks may be summarized in the following table:

| BENCHMARK PROBLEM | NASTRAN SOLID ELEMENT | | | |
|---|---|---|---|---|
| | HEXA1 | HEXA2 | IHEX1 | IHEX2 |
| 1. Cantilevered Beam (5 elements, 1 layer) | Poor | Poor | Poor | Good |
| 2. Thick-walled Cylinder (2,5,10 elements) | Good | Good | Good | Good |
| 3. Unit-cube Eigenvalues (1 element, consistent or lumped mass) | Poor | Fair | Good | (Not evaluated) |

In the eigenvalue extraction benchmark, we also found that the Givens or tridiagonal method was better than the inverse power method with shifts, and both were superior to the determinant method for this problem. (The FEER method was not used in this evaluation, but is expected to be equal to or better than Givens.)

The fairly good performance of the constant-strain HEXA2 element in our study corroborates with the conclusions reached in a recent Air Force Aero Propulsion Laboratory/NASA Lewis joint study on a three-dimensional stress analysis of a thermally-cycled double-edge wedge geometry specimen similar to those used in aircraft gas turbine engines (Ref. 52). Drake et. al., found excellent correlation for both displacements and stresses between a 354-element NASTRAN HEXA2 model and a 64-element model using a Garrett/Airesearch-developed code called ISO3DQ which used 12-node isoparametric solid elements.

## 7.8. Minicomputer Versions of NASTRAN

Minicomputers offer many advantages to the user and have increasingly gained popularity among structural analysis code users and developers. Among these advantages are: vastly lower cost per problem (sometimes by an order of magnitude or more); practical decentralization and greater flexibility; suitability for interactive graphics; economical time-sharing by many users (up to as many as 63 simultaneous users); shorter and predictable turnaround time; local control over computing resource allocation; and for the first time, a chance for the software developer to request operating system characteristics on minicomputers to be tailored to his needs. Typically, minicomputers cost about a quarter of a million dollars, contain virtual memories, and offer 32-bit architecture. Users of minicomputers usually rave about the reduced cost, versatility, turnaround, and efficiency.

Many U. S. minicomputer manufacturers compete for the market. The leader is Digital Equipment Corporation (DEC), with approximately 40% of the market (Ref. 4). Other well-known mini manufacturers include: Hewlett-Packard, Data General, Honeywell Information Systems, Prime, Modcomp and Perkin-Elmer. The 1979 revenues to U. S. minicomputer makers were estimated to be $4.7 billion, and in 1980 135,000 minis are expected to be shipped. Codes already on minis include: SAP6 and SAP7, SDRC/SUPERB and SUPERTAB, Westinghouse/WECAN, ANSYS, PDA/PATRAN, MARC/PM NASTRAN and MENTAT, and of course, versions of NASTRAN.

To the best of our knowledge, three minicomputer versions of NASTRAN currently exist:

| NAME | CODE DEVELOPER | MINI | MINI MANUFACTURER |
|---|---|---|---|
| MSC/NASTRAN | MacNeal-Schwendler Corp.<br>Los Angeles, CA | DEC VAX-11/780 | Digital Equipment<br>Marlboro, Mass. |
| UAI/NASTRAN | Universal Analytics Inc.<br>Playa del Rey, CA | Perkin-Elmer<br>3240 | Perkin-Elmer Corp.<br>Oceanport, N. J. |
| PM/NASTRAN | MARC Analysis Research Corp.<br>Palo Alto, CA | Prime 450,550,<br>650,750 | Prime Computer Inc.<br>Wellesley Hills,<br>Mass. |

The minicomputer version of MSC/NASTRAN is most commonly available on the DEC VAX-11/780, and thus VAX/NASTRAN is by far the most popular minicomputer version of NASTRAN around. DEC is also the leader in implementing structural analysis and design codes on minis. Its latest "Engineering Systems Software Referral Catalog" (5th edition, 1980) features no less than forty structural analysis and design packages available on various DEC minicomputers. UAI/NASTRAN and PM/NASTRAN are relative newcomers to the field. UAI/NASTRAN is being offered on the Perkin-Elmer 3240, the latest in the Series 3200, which reputedly performed faster than the DEC VAX-11/780 in 44 out of 58 benchmark tests. PM/NASTRAN is offered by MARC Analysis on Prime minicomputers, and its Version 80 features four new elements: HEXA3 and HEXA4 (8-node constant strain and 20-node linear strain isoparametric solid elements), and SHELL1 and SHELL2 (8-node linear strain and 12-node quadratic strain doubly-curved isoparametric shell elements). To our knowledge, there are no minicomputer versions of COSMIC/NASTRAN.

## 7.9. Preprocessors and Postprocessors for NASTRAN

One weakness of COSMIC/NASTRAN is its lack of a built-in interactive mesh generation capability. In the past five years, many software packages have been developed commercially to facilitate preprocessing and postprocessing for NASTRAN. These usually offer the user a capability to generate and modify the model in a time-sharing mode, and an instantaneous check of model connectivity on a cathode-ray-tube terminal screen. In addition, several U. S. government agencies and aerospace companies - such as the Naval Ship Research and Development Center, NASA Goddard Space Flight Center, and Rockwell International Corporation - have developed pre- and postprocessing packages (available from COSMIC) to aid the NASTRAN user in his model generation and results interpretation.

Here, only some of the more popular commercial NASTRAN pre- and post-processing packages will be discussed. These codes are typically available in minicomputer versions, and most are also available through the leading service bureaus. Table 7.9-1 shows some well-known commercial NASTRAN pre-and post-processors, their developers, marketing partner, and availability on service bureaus. The acknowledged leader in the field is SDRC/SUPERTAB, the first interactive preprocessing code to emerge. For finite element modeling and NASTRAN pre-processing, SUPERTAB has the most users and benefits from a marketing associative effort with Tektronix, the leader among CRT terminal manufacturers. FASTDRAW is offered by McAuto and supports finite element modeling for NASTRAN and other codes. GIFTS was developed at the University of Arizona under Professor H. Kamel; its popularity appears to have declined recently. A versatile preprocessor which is rapidly gaining in popularity is PDA/PATRAN, a powerful second-generation finite element modeling package which offers color interactive graphics. MENTAT and GRAFAX are less well-known packages which are beginning to build a user base. GENFEM is a popular European finite element code which reportedly can also generate NASTRAN models. In the European market, it competes with ASKA, ASAS, SESAM69, BERSAFE, FINEL, SAMCEF, TPS10, and PAFEC75 (Ref. 21). Preprocessors and minicomputers are here to stay. The NASTRAN user should take advantage of them because the future will bring more powerful packages, increased availability of minis and graphics terminals, and reduced costs in model generation and post-processing.

TABLE 7.9-1. COMMERCIAL NASTRAN PRE- AND POSTPROCESSOR SOFTWARE PACKAGES

| AME | DEVELOPER | MARKETED IN CONJUNCTION WITH | AVAILABILITY ON SERVICE BUREAUS |
|---|---|---|---|
| UPERTAB | Structural Dynamics Research Corp. Cincinnati, Ohio | Tektronix, Beaverton, Oregon<br>Digital Equipment Corp. Marlboro, Massachusetts | Control Data Corp. CYBERNET and others |
| ASTDRAW | McDonnell Douglas Automation Co., St. Louis, Missouri | McDonnell Douglas Automation Company | McAuto |
| ATRAN-G Version 1.2) | PDA Engineering Santa Ana, California | .Digital Equipment Corporation, Marlboro, Massachusetts (VAX-11/780)<br>.Atkins (United Kingdom)<br>.Rikei (Japan) | .United Computing Services<br>.Martin-Marietta Data Systems, Inc.<br>.University Computing Company |
| IFTS5 | University of Arizona, Tucson | Information Systems Design, Inc. Santa Clara, California | ISD |
| ENTAT | *MARC Analysis Research Corp.* Palo Alto, California | Prime Computer Inc., Wellesley Hills, Massachusetts | None. Currently used at MARC code installations |
| RAFAX | A. O. Smith Corp. Milwaukee, Wisconsin | | TELENET |
| EMSEN | FEGS LTD. Cambridge, England | - | Primarily on European service bureaus |

## 7.10. Program Efficiency Study

One of the stated objectives of this project was to evaluate the program efficiency. This is a formidable task for a general purpose program such as NASTRAN because of the complex nature of the program. For example, the mainframe version that is utilized (i.e., CDC, IBM, etc.) will certainly affect the results of such a study. In each version there are numerous differences, such as overlay structure, use of double precision arithmetic, type of loader, availability of central memory, etc., that can affect running time and efficiency. Even if the study is restricted to a particular machine, the program efficiency is an elusive commodity to assess. Not only are there a wide variety of rigid format solution procedures to be evaluated, but within the rigid formats there are often several options that may affect the efficiency, i.e., selection of eigenvalue extraction method. The use of DMAP and ALTER features will also have an effect (usually adverse) on the efficiency of NASTRAN. However, there are some general observations that can be made concerning program efficiency, some of which were learned during the course of this project and others obtained in the course of discussions with users.

It is generally accepted that NASTRAN was designed to solve large problems. Indeed, some of the problems that have been solved using NASTRAN probably could not have been solved using any other computer program. Clearly, program efficiency is not an influencing factor in such cases. The automated substructuring capability in NASTRAN may be particularly efficient for large problems in which parts of the geometry are repeated. This capability, which was recently added, is probably the most convenient substructuring method that is currently available, and requires a minimum of file handling by the user. Also, it is easy to use and can result in large savings in computer cost and engineering time.

It is generally felt that NASTRAN solves medium to large shell and beam structures as well or better than most general purpose programs. This is probably one of the strongest virtues of the code. Also, it is generally concluded that the rigid format solutions in NASTRAN have been well written and work efficiently. However, use of the ALTER or DMAP features can greatly

increase running times. As a particular example, one user reported that NASTRAN was being used for seismic analysis of piping systems. DMAP was used to calculate modal participation factors and to perform the modal superposition. Running times appeared to be almost an order of magnitude greater than would be expected using programs that are normally used in such analyses.

In order to obtain a quantitative measure of the efficiency of the program for one particular case, a series of runs were made in which a static analysis was performed of a flat plate with a central point load. (These runs were actually an extension of the convergence test described in Section 7.2.2 for QDPLT). The results of these runs are shown on Figure 7.10-1. The computer used was a CDC Cyber 175. The costs were obtained from the day file from each run and were computed using an internal algorithm that is used by the commercial data center (United Computing Systems) where the runs were made.

The largest problem run was just over 2000 degrees of freedom, which could be classified as a "moderate" size problem. The results of this study indicated that both the cost and CPU time increased approximately linearly with the number of degrees of freedom over the problem size range studied. A linear increase in cost versus problem size implies that the code will be relatively more efficient for larger problems. The time required to perform a matrix decomposition is a complicated function of machine timing constants and matrix topological parameters (see Ref. 13). However, an approximate formula for the time required for matrix decomposition is:

$$T = \frac{1}{2} M N W_{rms}^{2}$$

where M is an experimentally determined machine dependent parameter for a multiply-add loop, N is the order of the matrix and $W_{rms}$ is the root mean square wavefront. The test problem utilized a square array of elements and thus the rms wavefront increased (approximately) linearly with the number of degrees of freedom in the problem. Thus it would be expected that run times would exhibit a quadratic increase with the total number of degrees of freedom. The linear relationship between run time and degrees of freedom implies that the solution algorithm is more efficient for larger problems than for smaller ones. For a more complete discussion on estimates of run times and costs, see Section 14 of Ref. 11.

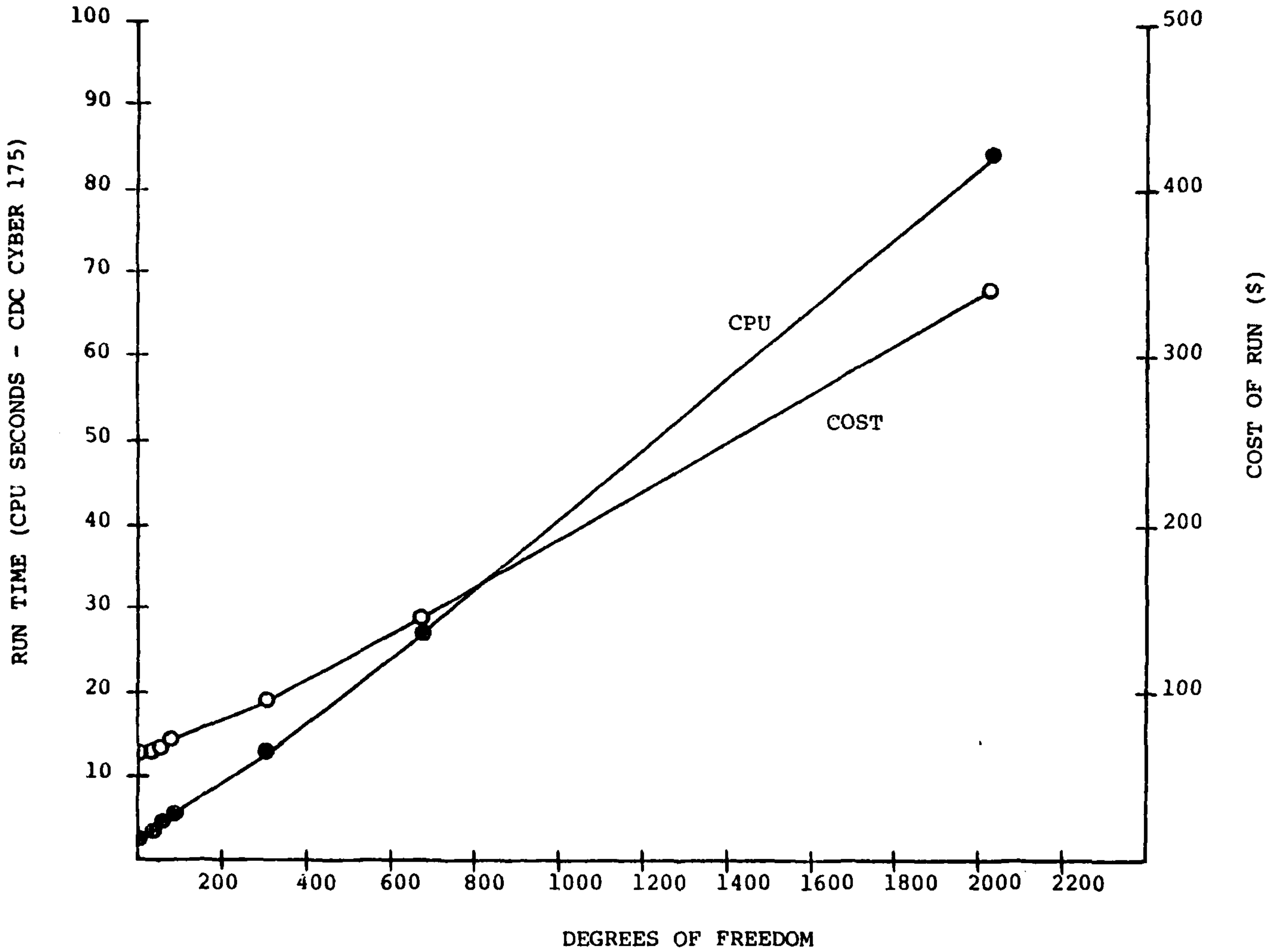



FIGURE 7.10-1. EFFECT OF PROBLEM SIZE ON COMPLETION TIME AND COST.

# SECTION 8

## CONCLUSIONS AND RECOMMENDATIONS

The results of this evaluation study of COSMIC/NASTRAN reveal that the code can be given an overall fair rating in its element library, capabilities, and efficiency. Documentation is rated good and in our opinion NASTRAN has the most complete documentation of any general purpose program. There are some areas where the documentation could be improved, but in general it is quite good. The strengths of NASTRAN are: the DMAP capability; versatility of analysis capabilities as offered in the 20 rigid formats; eigenvalue extraction options; plotting; restarting; and automated multi-stage substructuring analysis. Its weaknesses are: lack of a higher order 2-D solid-of-revolution element to handle plane strain, plane stress, or axisymmetric cases; user support and lack of a "hot-line" service; difficult-to-read documentation; lack of an adequate nonlinear analysis capability; a program architecture which is unsuitable for interactive use; and a time-consuming and ineffective error correction procedure. In the following subsections, an assessment of the program features and conclusions/recommendations will be covered separately.

### 8.1. Program Assessment

.Element Library

| Element | Performance Rating | Comments |
|---|---|---|
| QDPLT | Fair | Twist moment with one edge clamped caused problems; passed convergence tests; failed patch test. Inferior to many other available plate bending elements. |
| TRSHL | Poor | Convergence performance in Scordelis-Lo cylindrical roof benchmark inferior to other commercial shell elements. |
| TRIM6 | Good | Performed well in cantilevered beam benchmark in comparison to QDMEM and QDMEM1 elements. |
| TRPLT1 | Good | Compared well in simply-supported rectangular plate bending benchmark with QUAD1, Melosh, and Clough elements. |
| CONEAX | Good | Thermal loading capability is suspect. Otherwise, overall performance and versatility are good. |
| TRAPRG | Fair | Good performance in thick-walled cylinder benchmark, but restrictive in use due to trapezoidal element requirement. Needs plane strain and plane stress capabilities. |
| TRAPAX | Fair | Good performance in thick-walled cylinder benchmark, trapezoidal requirement; can handle nonaxisymmetric loads. |

8.1. (cont'd).

| Element | Performance Rating | Comments |
|---|---|---|
| HEXA1 | Poor | Poor performance in cantilevered beam benchmark and modal analysis; existence of spurious lateral displacement; adequate convergence in thick-walled cylinder benchmark. |
| HEXA2 | Fair | Poor rating in cantilevered beam benchmark; adequate performance in thick-walled cylinder and modal analysis. |
| IHEX1 | Fair | Surprisingly poor performance in cantilevered beam and modal analysis; good in thick-walled cylinder. May need "incompatible bending modes" to improve bending capability. |
| IHEX2 | Good | Overall good performance in cantilevered beam benchmark, thick-walled cylinder, and modal analysis. |

.Program Documentation

The manuals can be made more readable and user-oriented with simple examples, summary tables, application and usage hints, cost estimates, etc. The User's Guide (Ref. 11) is a good start.

.Error Correction Procedure and User Support

These areas were the subject of many user complaints. However, the nature of COSMIC/NASTRAN development history, implementation, and maintenance (Section 2) seems to make these areas difficult to change and improve. Recently implemented user support efforts at COSMIC appear to be improving in this respect.

.Program Architecture

The modular concept behind NASTRAN program architecture is flexible, but is based on computer core availability in the late sixties and is now difficult to adapt to a time-sharing system and interactive use.

.DMAP

The matrix handling capability and user control offered in DMAP are very flexible and unique among general purpose codes.

.Constitutive Library

Adequate, but deficient for general orthotropic or anisotropic analyses.

.Eigenvalue Extraction

COSMIC/NASTRAN's four eigenvalue extraction methods are the most offered in any code. All seem to work well, except the determinant method (see Subsection 7.7.2.3).

.Nonlinear Capabilities

COSMIC/NASTRAN offers limited nonlinear analysis capabilities, which are weak compared to many other nonlinear codes currently available.

.Special Features

Special NASTRAN features - such as static analysis with inertia relief, fully-stressed design optimization, cyclic symmetry, component mode synthesis, representation of part of a structure by its vibration modes, analyses of control systems, structure/fluid interaction, aeroelasticity, and acoustic cavities - are unique among current competing codes.

.Automated Multi-stage Substructuring Analysis

AMSS is flexible and powerful. This highly-developed substructuring capability is matched only by ANSYS among current structural analysis codes.

.Minicomputer Versions of COSMIC/NASTRAN

None, but three other minicomputer versions of NASTRAN exist: MSC/NASTRAN, UAI/NASTRAN, and PM/NASTRAN.

.Pre- and Post-Processors for NASTRAN

Many interactive graphics pre- and post-processors have been developed in the past five years for NASTRAN. Among these commercial packages are: SDRC/SUPERTAB, MCAUTO/FASTDRAW, PDA/PATRAN, GIFTS, MARC/MENTAT, and AOS/GRAFAX.

.Cost and CPU's versus Mesh Size

Both are approximately linear with mesh size (Fig. 7.10).

## 8.2. Conclusions and Recommendations

.NASTRAN program architecture is flexible, but has not kept pace with recent developments in computer hardware and operating system software. However, major changes in the architecture would be prohibitively expensive and would require a major rewriting of the program. The gains would probably not justify the expense.

. NASTRAN is designed for large problems. Small or moderate size problems are expensive compared to other general purpose codes. The lack of self-contained mesh generation capability almost necessitates the use of a pre-processor to develop a finite element mesh. This further reduces the usefulness of the program for small to medium size problems because of the additional step in converting the output of the mesh generation program into NASTRAN data, learning how to use the mesh generator, and the additional control cards. The novice and even the common user are often frustrated by this additional effort.

. No interactive capability exists. Many of the current general purpose codes can be executed in an interactive mode. It is expected that this trend will become more and more prevalent in the near future. The present program architecture of NASTRAN would be quite awkward to convert to an interactive mode.

. The documentation is bulky and not user oriented. It could and should be improved.

. The matrix handling package (DMAP) is very powerful. Better documentation and examples of its use would probably greatly benefit the novice and common user.

. To improve the efficiency of the finite element library and to reduce confusion to users (especially relatively new ones), some of the older and less effective elements (such as TRSHL, HEXA1, HEXA2) should be removed from the program unless they have unique capabilities that cannot be performed by other elements. Improved elements are needed in some areas (as discussed previously). The myriad of 13 separate membrane and bending elements in COSMIC/NASTRAN --TRMEM, TRIM6, QDMEM, QDMEM1, QDMEM2, TRPLT, TRPLT1, QDPLT, TRBSC, TRIA1, TRIA2, QUAD1, QUAD2 -- should be revised, consolidated and improved to offer the analyst fewer but better choices. A brief synopsis of each element, its capability and limitations should be included in the Users Manual so that reference to the Theoretical Manual is unnecessary for most analyses. Alternately, the Users Manual could describe the "recommended" elements and the experienced user would have access to the remaining elements if desired.

. It is our opinion that the only hexahedral elements offered should be isoparametric in formulation, with clearly documented hints on the merits of reduced integration and incompatible bending modes in various applications. The poor performance of IHEX1 on the cantilever beam problem could be greatly improved if incompatible bending mode formulation was added to this element. Otherwise, the 20-node

hexahedron IHEX2 should be used if only one element through the thickness is used when bending loads are present (for example, to represent thick shells). The additional complexity in modeling using a 20-node hexahedron versus an 8-node element results in increased engineering time and greater chance of modeling errors.

- NASTRAN is primarily a linear code. The nonlinear features are relatively few and not extensive.
- An organization which uses or intends to use the public domain versions of NASTRAN should be prepared to commit to a significant investment in staff for prompt support, consultation and system (job control) help. Our findings indicate that without this level of support, most organizations are frustrated by the lack of these necessary functions.
- Error correction in COSMIC/NASTRAN is time-consuming and ineffective. The time lag between when errors are reported and corrected is such that most analysts cannot wait for the response from the NASTRAN subcontractor. Many organizations which use NASTRAN correct errors as they are uncovered and may or may not report these changes to COSMIC or its subcontractor. This process results in many (mostly minor) variations of NASTRAN among the user community and uniform quality control is virtually impossible. It is incumbent upon the user or his organization to verify the accuracy of the program at each installation.
- In our opinion, NASTRAN is a program that should be maintained for public usage. While the program is in some respects relatively outdated, we feel that the program is still extremely versatile and serves a very useful purpose. One of the strongest arguments in favor of this is the fact that NASTRAN has been used by such a large number of analysts over the years. Such extensive usage is one of the best methods of verifying a program. Also, this large community of users is familiar with the program and its capabilities. This body of expertise is a very valuable asset. The high degree of modularity of the program will allow for the addition of new elements and new analysis procedures without adversely affecting the existing program. Verification of new features

is relatively simple compared to some other general purpose codes because of this modularity.

. The selection of a general purpose program for use by a company or organization must include the following factors: (1) Cost, (2) Capability (including element library and analysis methods), and (3) User Support. The cost of a program includes not only the initial cost of the source code (or the royalty fee for commercial programs) and computer time, but the cost of maintaining the program on the user's system. For example, implementing new features that are needed, error correction, and other maintenance functions can be time-consuming and expensive. Secondly, the capability of the program (Item 2) should not be static, but must change as new developments in element theory, analysis methods, etc. are developed. Thirdly, user support is very important. If possible, the principal developers of the program should be available for consultation when problems arise. It is extremely inefficient for even an expert user to find and correct programming errors, and this effort probably is impossible for most users. It is our opinion that all of these functions should be provided by a single organization, if possible. Such an organization is also needed to plan the future developments and additions that will be necessary if NASTRAN is to survive as a viable program.

# SECTION 9

## REFERENCES


1. Wilkes, M. V., "Computers into the 1980's", Electronics and Power, January 1980, pp. 67-71.

2. Scarrott, G. G., "From Computing Slave to Knowledgeable Servant: The Evolution of Computers", The Clifford Paterson Lecture (1979), Proc. Royal Society (London), Vol. 369A, 13 December 1979, pp. 1-30.

3. Mueller, G. E., "The Future of Data Processing in Aerospace", Aeronautical Journal, April 1979, pp. 149-158.

4. "Trends in Computing: Applications for the 80's", Fortune, 19 May 1980, pp. 29-70.

5. Sugarman, R., "Superpower Computers", IEEE Spectrum, April, 1980, pp. 28-34.

6. Darby, M. J., "The Expanding Bubble Market", Electronics Industry, February 1980, pp. 38-47.

7. The NASTRAN Theoretical Manual (Level 17.5) NASA SP221 (05) Dec. 1978 (COSMIC, 112 Barrow Hall, University of Georgia, Athens, GA 30602).

8. The NASTRAN User's Manual (Level 17.5) NASA SP222 (05) December 1978.

9. The NASTRAN Programmer's Manual (Level 17.5) SP223 (3 & 4) 1979.

10. The NASTRAN Demonstration Manual (Level 17.0) NASA SP224 (04) December 1977.

11. Field, E. I., D. N. Herting, and M. J. Morgan, NASTRAN User's Guide (Level 17.5), Prepared by Universal Analytics, Inc. (Playa Del Rey, California) for Langley Research Center, NASA CR-3146, June 1979.

12. McCormick, C. W., MSC/NASTRAN Basic Training Manual, MSR-46, 15 July 1977, (MacNeal-Schwendler Corporation, 7442 N. Figueroa Street, Los Angeles, CA 90041.)

13. Schaeffer, H. G., MSC/NASTRAN Primer: Static and Normal Modes Analysis, April 1979. Schaeffer Analysis, Inc., Kendall Hill Road, Mount Vernon, New Hampshire 03057.

14. A Brief Description of MSC/NASTRAN,MSR-54, April 1978 (MSC).

15. MARC General Purpose Finite Element Computer Program, User's Manual, Volumes A-G, 1979. MARC Analysis Research Corporation, 260 Sheridan Ave., Suite 200, Palo Alto, CA 94306.

16. Bathe, K. J., ADINA-A Finite Element Program for Automatic Dynamic Incremental Nonlinear Analysis, MIT Acoustics and Vibration Laboratory, Mechanical Engineering Department, Report 82448-1, 1977, Cambridge, Massachusetts.

17. Bathe, K. J., Static and Dynamic Geometric and Material Nonlinear Analysis Using ADINA, MIT Acoustics and Vibration Laboratory, Mechanical Engineering Department, Report 82448-2, 1977, Cambridge, Massachusetts.

18. Bathe, K. J., ADINAT - A Finite Element Program for Automatic Dynamic Incremental Nonlinear Analysis of Temperatures, MIT Acoustics and Vibration Laboratory, Mechanical Engineering Department, Report 82448-5, 1977, Cambridge, Massachusetts.

19. Haisler, W. E., AGGIE I-A Finite Element Program for Nonlinear Structural Analysis, Texas A&M University, Aerospace Engineering Department, Technical Report No. 3275-77-1, June 1977.

20. DeSalvo, G. J. and Swanson, J. A., ANSYS User's Manual, Swanson Analysis Systems, Inc., 1 July 1979, P. O. Box 65, Houston, Pennsylvania 15342.

21. Fredriksson, B. and Mackerle, J., "Overview and Evaluation of Some Versatile General Purpose Finite Element Computer Programs", Second World Congress on Finite Element Methods, 23-27 October 1978, Bournemouth, England.

22. Dunder, V. F. and Belonogoff, G., "Comparing Finite Element Programs in Engineering," Conference on Finite Element Methods and Engineering, sponsored by the MacNeal-Schwendler Corporation, 13-14 March, 1980, Pasadena, CA.

23. Rosen, R., STARDYNE Theoretical and User Information Manuals, Control Data Corporation, Publication Nos. 86616300 (1/25/74) and 76079900 (4/1/78).

24. SAP6 User's Manual - "A Structural Analysis Program for Static and Dynamic Analysis", SAP User's Group, University of Southern California, Civil Engineering Department, November 1977.

25. Blissmer, F. P., Kirby, R. W., and Peterson, F. E., EASE2-Elastic Analysis for Structural Engineering, Engineering Analysis Corporation, 1979.

26. Nicolas, V. T., and Citipitioglu, E., "A General Isoparametric Finite Element Program: SDRC SUPERB", Int'l. J. of Computers and Structures, Vol. 7, No. 2, pp. 303-314, 1977.

27. Field, E. I., Herting, D. N., Herendeen, D. L., and Hoesly, R. L., "The Automater Multi-stage Substructuring System for NASTRAN", NASTRAN: User's Experiences, NASA TM X-3278, Fourth NASTRAN Users' Colloquium, NASA Langley Research Center, 9-11 September, 1975.

28. Herting, D. N., and Hoesly, R. L., "Development of An Automated Multi-stage Modal Synthesis System for NASTRAN," Sixth NASTRAN Users' Colloquium, NASA Conference Publication 2018, pp. 435-448, Lewis Research Center, 4-6 October 1977.

29. Herting, D. N., and Morgan, M. J., "A General Purpose, Multi-stage, Component Modal Synthesis Method", 20th AIAA/ASME/ASCE/AHS Structures, Structural Dynamics and Materials Conference, 6 April 1979, St. Louis, Missouri.

30. DeSalvo, G. J., ANSYS Verification Manual, Swanson Analysis Systems, Inc., 1 April, 1979, Box 65, Houston, Pennsylvania, 15342.

31. Robinson, J. "Element Evaluation-A Set of Assessment Points and Standard Tests", Second World Congress on Finite Element Methods, 23-27 October 1978, Bournemouth, England, (Published as Finite Element Methods in the Commercial Environment, ed. J. Robinson, Robinson and Associates, Horton Road, Woodlands, Wimborne, Dorset BH21 6NB, England).

32. Robinson, J. "A Single Element Test", Computer Methods in Applied Mechanics and Engineering, Vol. 7, 1976, pp. 191-200.

33. Robinson, J. and Haggenmacher, G. W., "LORA - An Accurate Four Node Stress Plate Bending Element", Int'l. J. Numercial Methods in Engineering, Vol. 14, pp. 287-312, 1979.

34. Robinson, J., and Blackham, S., "An Evaluation of Lower Order Membranes As Contained in the MSC/NASTRAN, ASAS, and PAFEC FEM Systems", Conference on Finite Element Methods and Technology (sponsored by MSC), Pasadena, CA, 13-14 March 1980.

35. Padovan, J. and Chang, T. Y.,"Evaluation of ADINA: Part I (Theory and Programming Descriptions);Part II (Operating Characteristics)", University of Akron, Akron, Ohio 44325, June 8, 1980.

36. Roark, R. J. and Young, W. C., Formulas for Stress and Strain, 5th ed., 1975, pp. 386, 392.

37. Narayanaswami, R.,"New Triangular Shallow Shell Finite Element," *Old Dominion* University, Technical Report 74-T7, November 1974.

38. Narayanaswami, R. "Addition of Higher Order Plate and Shell Elements into NASTRAN Computer Program", Old Dominion University, Technical Report 76-719, December 1976.

39. Potvin, A. B., Exxon Production Research Company, private communication, March 10, 1980.

40. Morgan, M. J., Field, E. I., and Herting, D. N., "Recent Enhancements to COSMIC/NASTRAN", SAP User's Conference, University of Southern California, June, 1978.

41. MacNeal, R. H.,"MSC/NASTRAN Status Report", Conference on Finite Element Methods and Technology (sponsored by MSC), March 13-14, 1980, Pasadena, California.

42. Potvin, A. B. and Leick, R. D., "NASTRAN Implementation of an Isoparametric Doubly-curved Quadrilateral Shell Element", Seventh NASTRAN Users Colloquium, Marshall Space Flight Center, October 4-6, 1978, NASA CP-2062, pp. 147-191.

43. Heller, P., MARC Analysis Research Corporation, private communication on pre-release PM/NASTRAN Version 80 results, July 18, 1980.

44. Grafton, P. E. and Strome, D. R., "Analysis of Axisymmetric Shells by the Direct Stiffness Method", J. AIAA, Vol. 1, No. 10, October 1963, pp. 2342-2347.

45. Jones, R. E. and Strome, D. R., "A Survey of Analysis of Shells by the Displacement Method", Proc. First Conference on Matrix Methods in Structural Mechanics, Wright-Patterson Air Force Base, Ohio, 26-28 October 1965, AFFDL-TR-66-30 (AD 646300), pp. 205-229.

46. Hetenyi, M., Beams on Elastic Foundation, University of Michigan Press, 1946.

47. Timoshenko, S. P., Strength of Materials (Vol. II), 3rd ed., Van Nostrand, 1956.

48. Clough, R. W., "Comparison of Three Dimensional Finite Elements", Proc. of Symposium of Application of Finite Element Methods in Civil Engineering, sponsored by the American Society of Civil Engineers, Vanderbilt University, Nashville, Tennessee, November 1969, pp. 1-26.

49. Zienkiewicz, O. C., The Finite Element Method, McGraw-Hill (U. K.), 1977, Chapter 6.

50. Gallagher, R. H., Finite Element Analysis - Fundamentals, Prentice-Hall, 1975, Chapter 10.

51. Bathe, K. J. and Wilson, E. L., Numerical Methods in Finite Element Analysis, Prentice-Hall, 1976.

52. Drake, S. K., et. al., "Three-Dimensional Finite Element Elastic Analysis of a Thermally Cycled Double-Edge Wedge Geometry Specimen", Air Force Wright Aeronautical Laboratories, Technical Report AFWAL-TR-80-2013, March 1980.

53. MSC/NASTRAN, Application Manual (Version 60), February 1980.

54. Desai, C. S. and Abel, J. F., Introduction to the Finite Element Method, Van Nostrand Reinhold, 1972, pp. 166-169.

55. Wilson, E. L., Taylor, R. L., Doherty, W. P., and Ghaboussi, J.,"Incompatible Displacement Models", Numerical and Computer Methods in Structural Mechanics, (ed. S. J. Fenves, et. al.), Academic Press, 1973.

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

# REPORT DOCUMENTATION PAGE

READ INSTRUCTIONS BEFORE COMPLETING FORM

| | | |
|---|---|---|
| 1. REPORT NUMBER<br>SSC-81980 | 2. GOVT ACCESSION NO.<br>AD-A092 210 | 3. RECIPIENT'S CATALOG NUMBER |
| 4. TITLE (and Subtitle)<br>Evaluation of the NASTRAN General Purpose Computer Program | | 5. TYPE OF REPORT & PERIOD COVERED<br>Technical Report |
| | | 6. PERFORMING ORG. REPORT NUMBER |
| 7. AUTHOR(s)<br>J. W. Jones<br>H. H. Fong<br>D. A. Blehm | | 8. CONTRACT OR GRANT NUMBER(s)<br>N00014-79-C-0576 |
| 9. PERFORMING ORGANIZATION NAME AND ADDRESS<br>Swanson Service Corporation<br>P. O. Box 5415<br>Huntington Beach, CA 92646 | | 10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS |
| 11. CONTROLLING OFFICE NAME AND ADDRESS<br>Office of Naval Research<br>Arlington, VA | | 12. REPORT DATE<br>August, 1980 |
| | | 13. NUMBER OF PAGES<br>135 |
| 14. MONITORING AGENCY NAME & ADDRESS(if different from Controlling Office) | | 15. SECURITY CLASS. (of this report)<br>Unclassified |
| | | 15a. DECLASSIFICATION/DOWNGRADING SCHEDULE |

16. DISTRIBUTION STATEMENT (of this Report)

Distribution of this document is unlimited.

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

18. SUPPLEMENTARY NOTES

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

NASTRAN, Finite Element, Structural Analysis, Computer Program

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

An evaluation of the NASTRAN general purpose computer program was made. The report discusses the current status (usage, documentation, contents) of public domain (COSMIC) versions of NASTRAN. The program architecture is evaluated and compared to other general purpose programs. The functional description of the program is discussed with respect to its capabilities and limitations. Advanced evaluation exercises are reported which give the results of a number of problems used to test various structural elements and solution techniques.

DD FORM 1 JAN 73 1473 EDITION OF 1 NOV 65 IS OBSOLETE
S/N 0102-014-6601 |

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)